THIS PAGE HAS BEEN LEFT INTENTIONALLY BLANK

NAMOI CATCHMENT WATER STUDY

INDEPENDENT EXPERT

MODEL USER MANUAL

July 2012

50371/P3-R3 FINAL

Prepared for:

Department of Trade and Investment, Regional Infrastructure and Services,
New South Wales, (DTIRIS NSW)
Locked Bag 21
Orange
NSW 2800

Prepared by:

Schlumberger Water Services (Australia) Pty Ltd
Level 5, 256 St Georges Terrace
Perth WA 6000
Australia

REPORT REVIEW SHEET

Namoi Catchment Water Study Independent Expert Model User Manual			
Report no:	50371/P3-R3 FINAL		
Client name:	Department of Trade and Investment, Regional Infrastructure and Services, New South Wales, (DTIRIS NSW)		
Client contact:	Mr Mal Peters, Chairman, Ministerial Oversight Committee (MOC)		
SWS Project Manager:	Sean Murphy		
SWS Technical Reviewer:	Mark Anderson		
Main Author:	Gareth Price		
Date	**Issue No**	**Revision**	**SWS Approval**
17/02/2012	1	Draft	Mark Anderson
27/07/2012	2	Final	Mark Anderson

CONTENTS

THIS PAGE HAS BEEN LEFT BLANK INTENTIONALLY

1 INTRODUCTION

Schlumberger Water Services (Australia) Pty Ltd (SWS) has been appointed as the Independent Expert for the Namoi Catchment Water Study (the Study) and has been charged with the development of an integrated suite of models (the Model) for the assessment of the nature and extent of potential effects from coal and gas developments on the water resources of the catchment. The purpose of the Study is to collate and analyse quality data to assist in identifying and quantifying risks associated with the coal mining and coal seam gas (CSG) developments on water resources.

The numerical modelling tools were developed as part of Phase 3 of the Study. Two numerical models were constructed that together have the ability to simulate those parts of the hydrological system that are pertinent to the simulation of the interaction between coal and gas development and the surface water and groundwater resources of the Namoi catchment. The two models constitute the Model and are:

- A lumped parameter Hydrologic Model. This was constructed with the LASCAM (Viney and Sivapalan, 2000) package and is used to simulate the fate of rainfall in the catchment, particularly the portion that forms runoff and the portion that percolates downwards into the sediments and recharges the groundwater system. This model includes the simulation of the impact of mining and CSG development on these processes.

- A groundwater flow model (the Groundwater Model). This was constructed using the numerical code MODFLOW 2000 (Harbaugh et al, 2000) and is used to simulate the processes governing groundwater flow in and between the alluvial aquifers and hydrostratigraphic units pertinent to the prediction of the impact of coal and gas developments on the groundwater resources and the interaction between surface water and groundwater. The model therefore includes representation of the abstraction of groundwater associated with CSG development and the flow of groundwater to mine voids, both underground and open cut. It also uses predictions from the Hydrologic Model to define groundwater recharge inputs and changes to these in response to mining and CSG development.

According to the Study Request for Tender (Section G.3.7.1.g.ii) a User Manual must be produced that provides full details on how to operate the Model including updating the Model to include additional or improved data and the locations for mines or gas extraction developments.

This document represents the User Manual. Section 2 provides details of the operation of the Groundwater Model and Section 3 the operation of the Hydrologic Model.

2 GROUNDWATER MODEL

2.1 Introduction

The MODFLOW 2000 numerical code (Harbaugh et al, 2000) along with the user interface Groundwater Vistas, Version 6 (ESI, 2011) is used to simulate groundwater flow and surface water / groundwater interaction in the Groundwater Model.

The modelling has been undertaken assuming saturated, single phase, temperature independent and single density groundwater flow.

2.2 Groundwater Vistas files

Nine Groundwater Vistas files and a single results file have been provided. They are:

- Namoi_Historical.gwv
- Namoi_Historical.hds
- Namoi_Sc0.gwv
- Namoi_Sc1.gwv
- Namoi_Sc2.gwv
- Namoi_Sc3.gwv
- Namoi_Sc4.gwv
- Namoi_Sc5.gwv
- Namoi_Sc6.gwv
- Namoi_Sc7.gwv

These files contain all of the data required to run the historical and predictive scenario models. The results file "Namoi_Historical.hds" forms the initial heads for all of the scenario runs. When running any of these scenarios for the first time the link to the "Namoi_Historical.hds" file must be defined within Groundwater Vistas.

2.3 Domain, grid and rotation

The model domain comprises a rectangular model grid, rotated by 30 degrees clockwise from north. This aligns the model cells with the Basin morphology, as the general dip direction of the hard rock units and orientation of the Upper Namoi Alluvium is along this axis.

The specific model properties are detailed below:

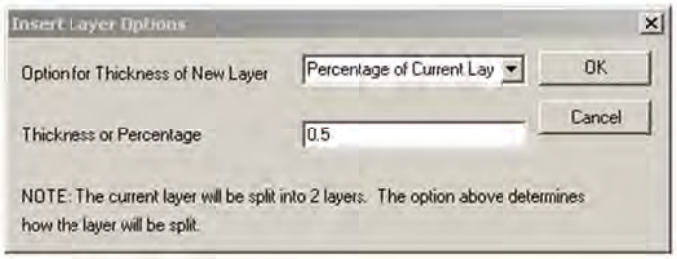
- The model origin is at 745,000E, 6,410,000N (MGA Zone 55, GDA94).
- The model extends 310 km to the NW and 180 km to the NE.
- The model cell size (plan view) is 1,000 m by 1,000 m (310 rows and 180 columns).
- The model contains 20 vertical layers, with thicknesses provided by the geological model described in Section 2.

Each model layer is composed of 55,800 cells. There are 20 model layers therefore resulting in a total number of cells for the model of 1,116,000.

2.4 Layer surfaces

Tasks involving the setup of layers and vertical discretization are mostly performed using the ***Grid*** and ***Props*** menus.

New layers can be added using the menu ***Grid > Insert > Layer above***, if the new layer is to be added above the current layer, or ***Grid > Insert > Layer below***, if the new layer is to be added below the current layer. In either case, the following dialog will appear:



The combo box **Option for Thickness of New Layer** provides options for inserting layers, the first one (**Percentage of Current Layer**) splits the current layer in two assigning a percentage of the current layer thickness to the new layer. The percentage is defined in the text box **Thickness or Percentage**.

The second option, **Constant thickness**, assigns a constant thickness for the new layer. The thickness of the new layer can be specified in the **Thickness or Percentage** text box.

Layers can be deleted in the menu ***Grid > Delete > Current layer***.

The thickness of layers can be edited in the menu ***Props > Top elevation*** or ***Props > Bottom elevation***, where the top and bottom elevations of layers can be edited respectively. Elevations can be assigned manually or imported via the menu ***Props > Import...*** where several different formats can be used.

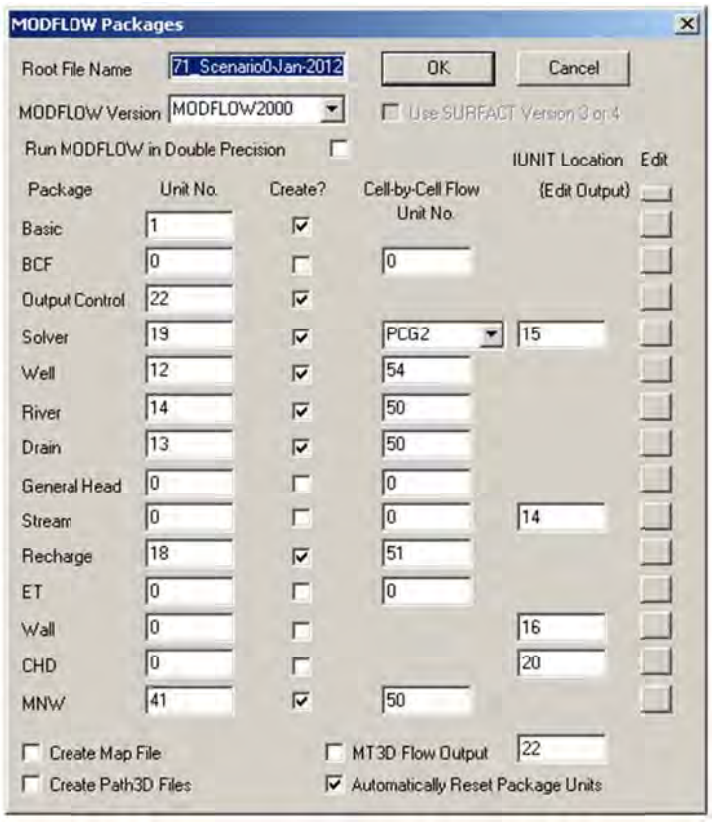
It is important to note that assigning the bottom elevation of one layer does not automatically assign the top elevation of the layer below, requiring that both top and bottom elevations are assigned for all layers. The same is valid for top elevation and bottom elevation of the layer above.

2.5 MODFLOW settings

2.5.1 *BCF / LPF*

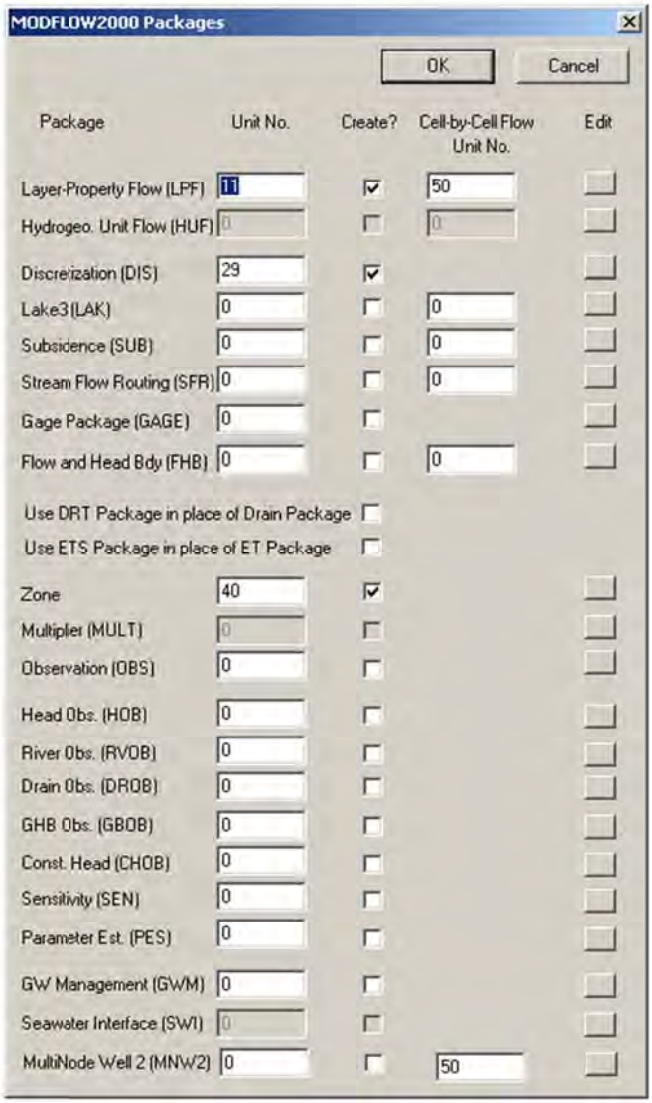
The choice between the Block Centred Flow (BCF) and Layer Property Flow (LPF) packages affects the way MODFLOW formulates interblock conductances and confined/unconfined behaviour.

The BCF package can be activated/deactivated using the Menu ***Model > MODFLOW > Packages...*** The following dialog will appear:



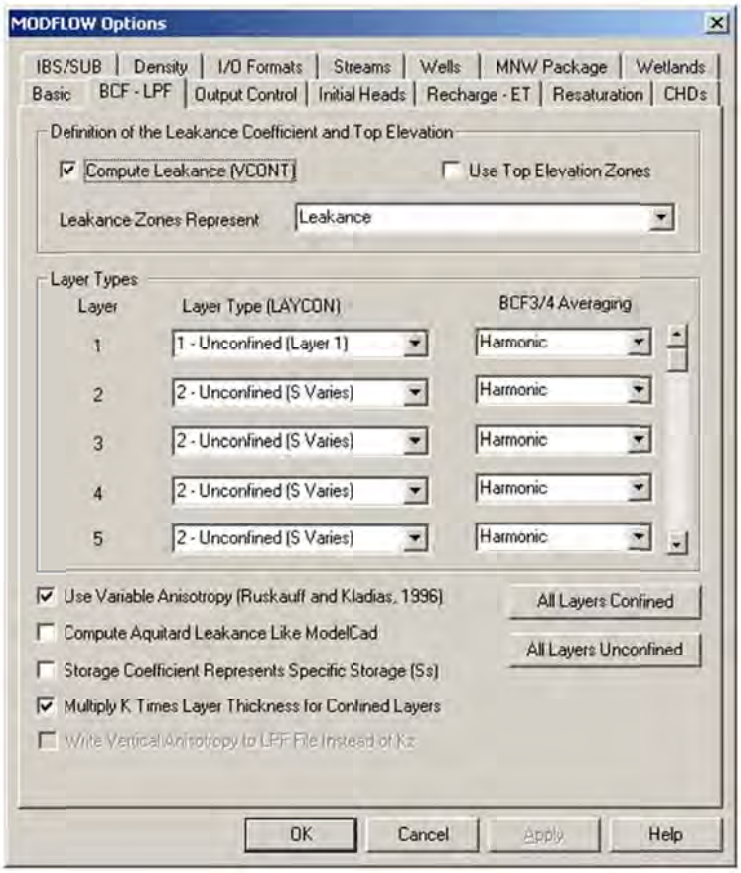
To activate the BCF package, click the check box next to the BCF package (2nd line).

The LPF package can be activated/deactivated through the menu ***Model > MODFLOW 2000 > Packages...*** A new dialog will appear (see below) and the package can be toggled by clicking on the check box next to the LPF line.



Important: Make sure that only one (LPF or BCF) package is activated. MODFLOW will not work when both or neither of the two packages are activated.

Specific layer parameters such as averaging, confined/unconfined behaviour and vertical leakance calculations can be modified using the menu ***Model > MODFLOW > Packages options...*** A new dialog will appear and the parameters can be edited in the BCF-LPF tab, illustrated below.



Parameters regarding vertical leakance can be found in the top box. Checking the **Compute Leakance (VCONT)** box makes MODFLOW calculate the vertical leakance based on cell thicknesses and vertical hydraulic conductivities as specified in the model (this is the option used in the Namoi Model).

If this option is not selected, leakance values will be assigned from the Leakance property, which can be represented as raw leakance values, vertical conductivity of the layer, vertical conductivity of the aquitard or vertical anisotropy. The leakance format can be chosen in the **Leakance Zones Represent** combo box.

Layer confined/unconfined behaviour can be defined in the **Layer Types** box. For each layer defined in the model, there will be a combo box in the column **Layer Type (LAYCON**). Layer type options available will depend on the selected flow package (BCF or LPF).

Interblock conductance averaging options can be defined in the column **BCF3/4 Averaging**. Averaging options can be individually assigned for each layer through the combo boxes. Averaging options also depend on the selected flow package.

2.5.2 *Initial heads*

The initial heads can be assigned in three different ways:

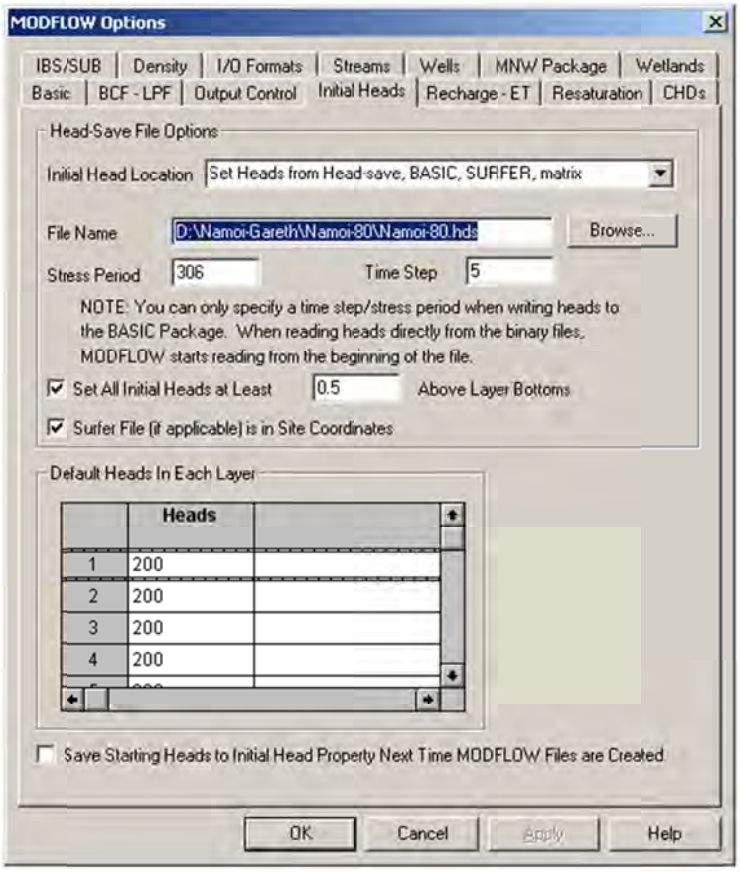
- Using constant values for each layer;
- Assigning values manually in the property editing mode; and
- Importing simulated heads from a previous MODFLOW row.

The three options can be found in the menu **Model > MODFLOW > Package Options...** . The **Modflow Options** appear and initial heads settings are found in the **Initial Heads** tab, as illustrated in the next figure.

The way initial heads are defined is chosen in the **Initial Head Location** combo box. Selecting the option **Use Default Heads in Spreadsheet Below** allows the use of one initial value per layer, defined in the table **Default heads in Each Layer** located at the bottom of the dialog.

The option **Set Heads from Head-save, BASIC, SURFER, matrix** allows the use of previously simulated heads from another model. This option is useful, for instance, when hydraulic heads from the historical model need to be used as initial heads for the predictive runs, and this is the way it is done in the Namoi model. The file containing the previously simulated heads can be defined in the **File Name** box and the desired time can be defined in the **Stress Period** and **Time Step** text boxes.

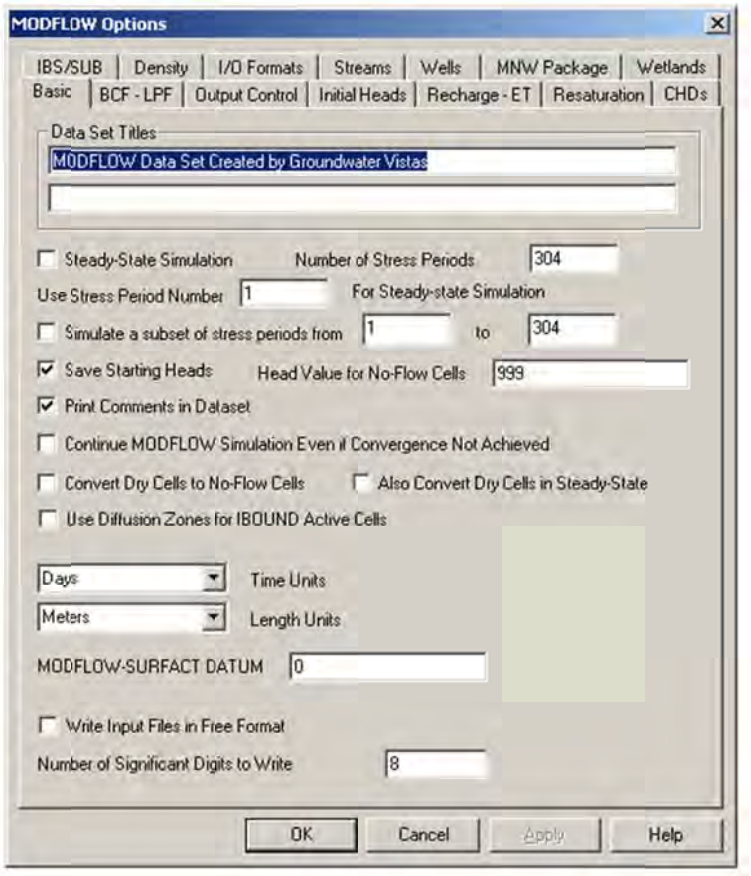
The option **Use Initial Heads Property Data** allows the manual editing of initial heads in the property editing mode, described in Section 2.8.



2.6 Time settings

MODFLOW models have their simulation period divided into stress periods and time steps. While stress periods define time periods in which boundary conditions do not change (e.g. recharge rates, abstraction volumes and river levels), time steps are subdivisions of the stress periods implemented in order to facilitate numerical convergence and increase the stability of the numerical solution.

Stress periods can be added or deleted using the menu ***Model > MODFLOW > Package options...*** The number of stress periods can be modified in the text box **Number of Stress Periods** located below the **Data Set Titles** box.

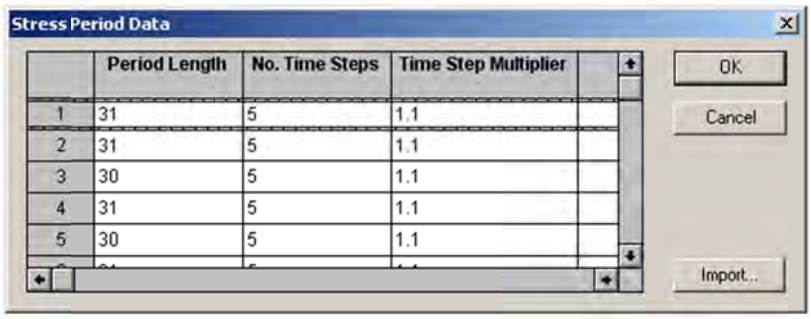


Modifications in the number of stress periods using the **MODFLOW Options** will be observed in the end of the simulation. Addition of stress periods will result in appending them at the end of the last stress period, while deletion of stress periods will be done from the last stress period backwards.

Stress periods can be inserted or deleted in the middle of the simulation, as opposed to the previous method which affect only the final stress period, using the menu ***Model > MODFLOW > Insert/delete Stress Periods...*** A new dialog appears (see below) where 3 operations can be performed:

- Delete starting with Stress Period
- Insert after Stress Period
- Insert before Stress period

Stress period length and time stepping options can be found in the menu ***Model > MODFLOW > Stress period Setup...*** A new dialog with a table containing the details of each stress period is activated, as illustrated in the figure below.



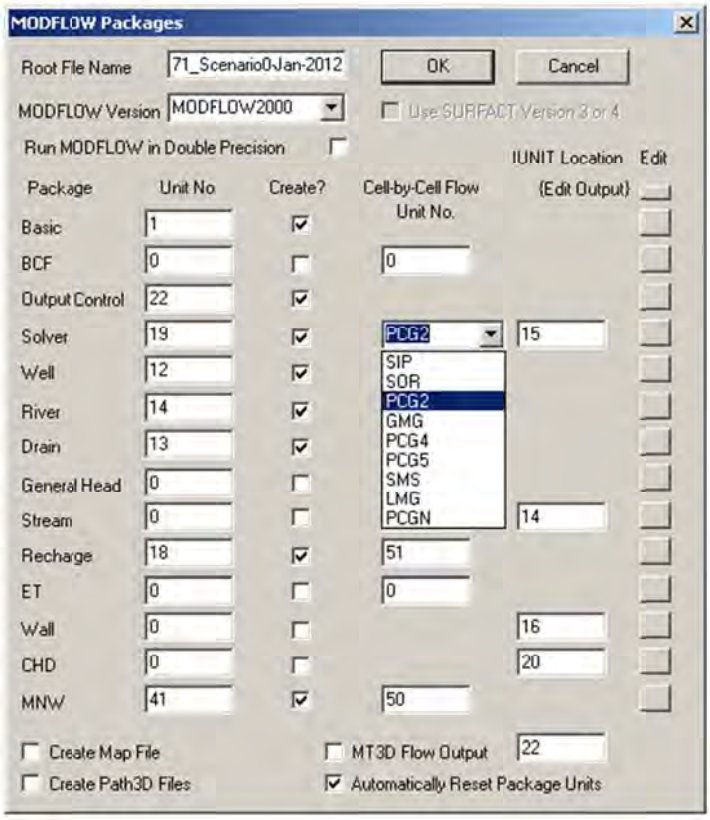
The table presents three columns as follows:

- **Period Length –** which defines de stress period length (defined in days in Namoi model)
- **No. Time Steps –** which defines the number of time steps for each stress period
- **Time Step Multiplier –** which defines the time step multiplier within the stress period. The value of 1 equates to having equal time step lengths. The value used in this model is 1.1.

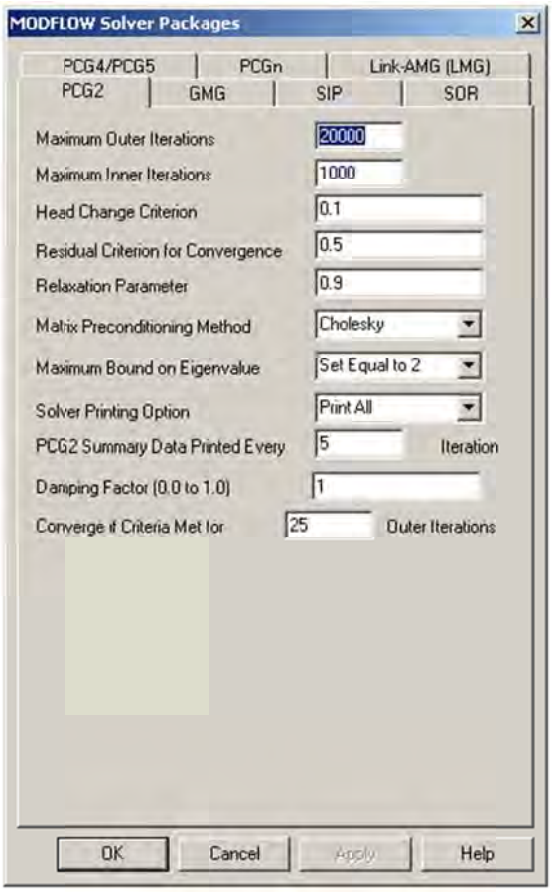
2.7 Solver settings

MODFLOW contains many solvers which can be used to solve the differential equations formulated for a given model. The choice of solver relies on the type and complexity of models, with PCG2 being the most commonly used.

The solver can be chosen through the menu ***Model > MODFLOW > Packages...*** . There is a combo box next to the solver package line, which determines which solver will be used (see below).



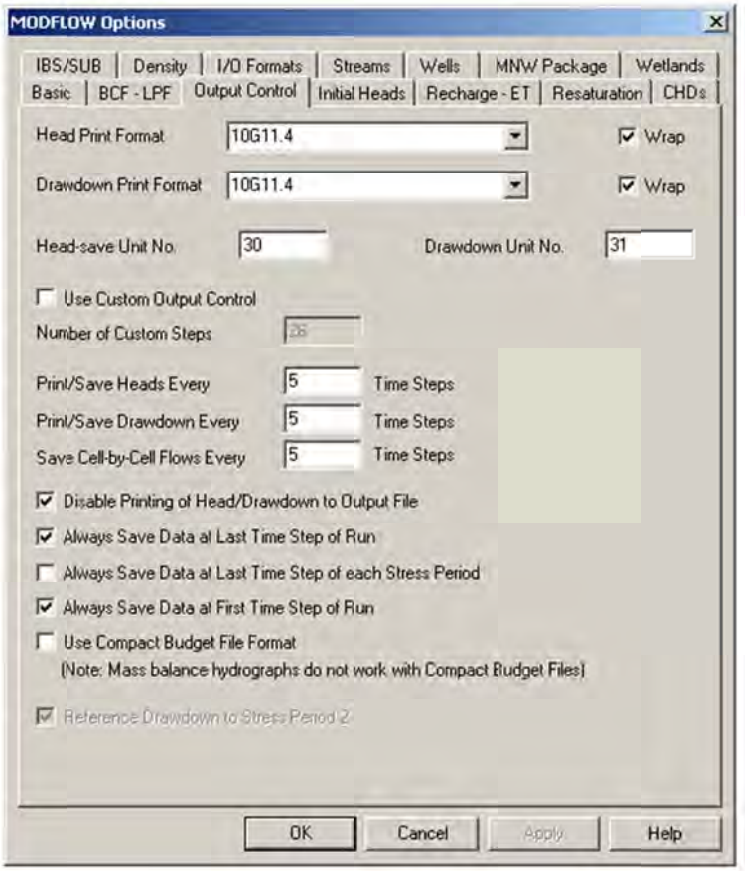
Solver settings can be changed using the menu ***Model > MODFLOW > Solver Options...*** . A new dialog with the settings for the several solvers will appear and is illustrated below:



Each tab on the dialog corresponds to the settings of a specific solver. The solver PCG2 was the solver used in the Namoi model, although other solvers can be used. The solver PCG4/PCG5 cannot be used since this solver is only present in the MODFLOW-SURFACT model. The LMG solver is proprietary and must be purchased prior to being used.

2.8 Output settings

The default output settings dialog can be accessed through the menu ***Model > MODFLOW > Package Options...*** in the **Output Control** tab.



The output frequency for hydraulic heads, drawdown and cell-by-cell flows can be set using the text boxes:

- **Print/Save Heads Every**
- **Print/Save Drawdown Every**
- **Print/Save Cell-by-Cell Flows Every**

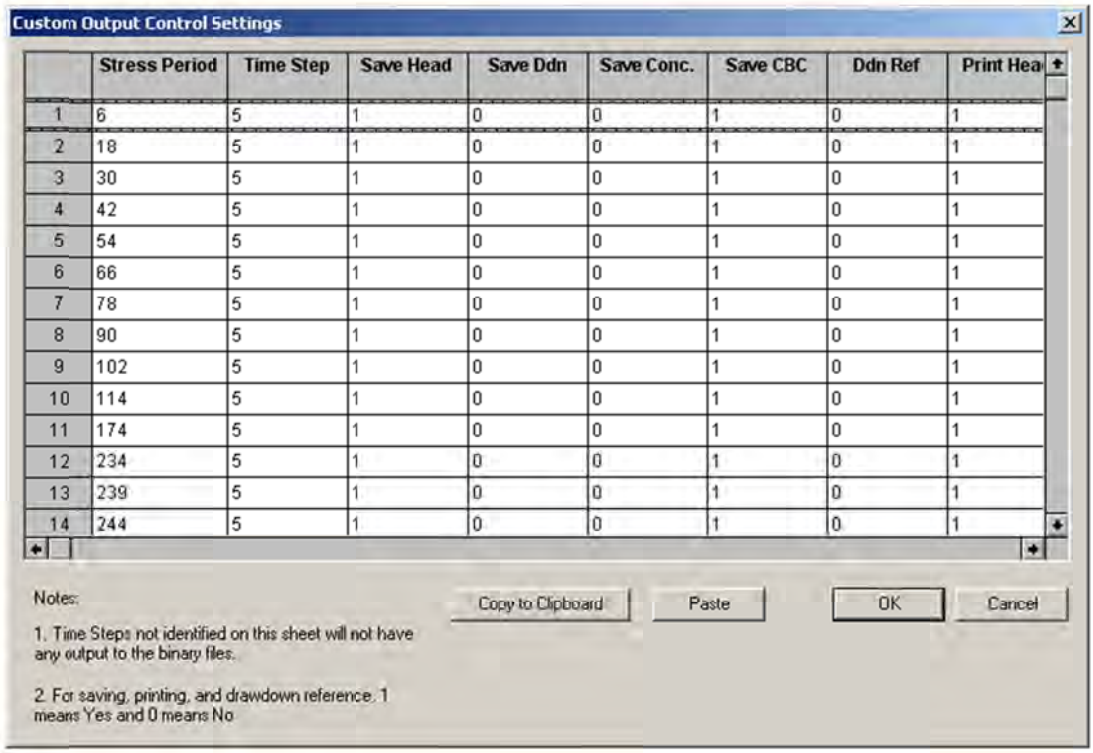
Outputs from the beginning and end of the simulation can be saved by marking the check boxes **Always Save Data at Last Time Step of Run** and **Always Save Data at First Time of Run**. Outputs from the end of each stress period can be saved marking the check box **Always Save Data at Last Time Step of each Stress Period**.

Simulation outputs can often be too large in terms of storage space, leading to slow reading times and difficult post processing. In order to overcome that, custom output can be defined in MODFLOW.

Custom outputs can be generated ticking the checkbox **Use Custom Output Control**. Custom output settings can be defined in the menu ***Model > MODFLOW > Custom Output Control...*** A dialog with a table of settings (see below) is displayed with the following main columns:

- **Stress Period** – specifies the stress period for the custom output.
- **Time Step** – specifies the time step for custom output
- **Save Head** – specifies if heads will be saved (1 = yes, 0 = no)
- **Save Ddn** – specifies if drawdown will be saved (1 = yes, 0 = no)
- **Save Conc.** – specifies if concentration will be saved. Not relevant to the Namoi Model
- **Save CBC** – specifies if cell-by-cell flows (for water balance calculations) will be saved (1 = yes, 0 = no)

The remaining columns refer to output to the MODFLOW list file and are not relevant to the Namoi Model.



2.9 Parameterisation

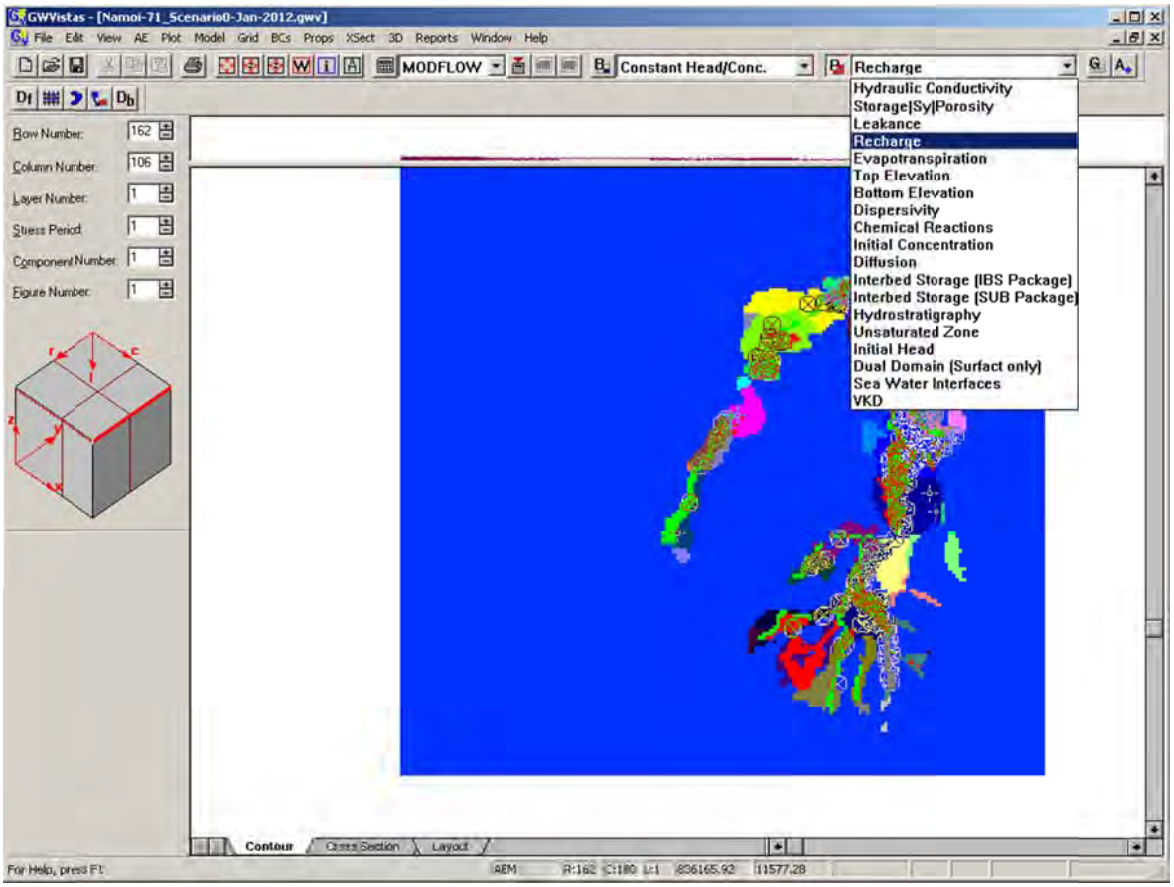
The majority of spatially-distributed parameters of the model can be accessed and modified in the property editing mode. This mode allows the displaying and editing of all properties. Properties relevant to the Namoi Model are:

- Hydraulic Conductivity;
- Storage/Porosity;
- Recharge;
- Top Elevation;
- Bottom Elevation; and
- Initial Heads.

Properties can be edited using zones of constant value or as matrices of continuous values. The dialog found in the menu ***Props > Property Options...*** opens the dialog for setting the mode of every property. By default, hydraulic conductivity, storage and recharge are treated with zone distributions, while layer elevations and initial heads are defined by continuous matrices.

The property zones distribution can be edited using the editing mode by pressing the  button.

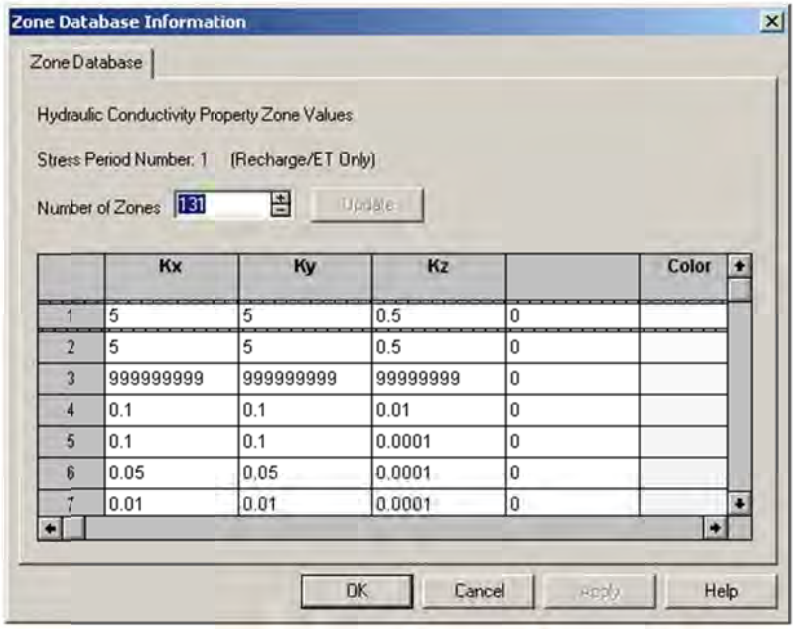
The property to be edited can be selected in the combo box next to the  button.



Once the property has been selected, the following options can be accessed from the toolbar or menu ***Props > Set Value or Zone***.

- This button activates the property editing mode
- This button defines the default zone to be defined by the editing commands
- This button is used to assign a rectangular zone
- This button is used to assign a polygon zone.
- This button transposes a property zone. The same can be done using the menu ***Props > Set Value or Zone > Transpose...***

Zone values for each property can be accessed through the Zone Database Information Dialog (button), where a table including zone numbers and respective parameter values are shown and can be edited (see figure below).



2.10 Boundary conditions

2.10.1 *Recharge*

Recharge zones were assigned in the Namoi Model to match the sub-catchment structure defined in the LASCAM model, so that infiltration values from LASCAM could be assigned directly to MODFLOW. The distribution of recharge zones can be edited in the property editing mode previously described. Specific zone values can be accessed and edited in the **Zone Database Information** dialog.

Daily LASCAM output values were converted to a monthly MODFLOW format using a specially written script. Monthly historical recharge from LASCAM was factored to an average catchment wide recharge value of 20 mm/yr to match with the recharge assigned to the calibrated and accepted Upper Namoi groundwater model (McNeilage, 2006). The factoring process was completed in Excel, with the steps as follows:

1. Remove any negative recharge values and replace them with a zero value.
2. Convert all of the recharge depths to a volume per month by multiplying by the sub-catchment area and the number of days in the month
3. Sum all of the sub-catchment volumetric recharges for each year and divide the total by the total area of the Hydrologic Model then 365 to get an average recharge depth per year (in mm/yr)
4. Calculate the average annual recharge over the years calculated (1990 - 2009)
5. Repeat steps 2 to 4 using a multiplication factor on the initial recharge depths until the average annual recharge equals 20 mm/yr. In the current model this required a factoring value of x 0.225
6. Output these new recharge values for use in the historical MODFLOW model

Recharge calculations for the scenario and sensitivity models were factored by this same value so that any changes to recharge would only be from changes to model inputs. Average recharge was therefore allowed to become higher or lower than 20 mm/yr depending on the scenario being run. Three zones showed anomalous recharge values in the LASCAM files and these were given fixed recharge values of 20 mm/yr (Zones 48 and 52), and 29.2 mm/yr (Zone 38). These values were then allowed to change in the same way as other zones with mining or CSG developments.

Open-cut mines were introduced to the predictive models by reducing the size of the historical sub-catchment by the maximum footprint area of the mine from the start of the simulation. Recharge was factored by 0.225 and then also factored by the new (reduced) catchment area.

For the predictive models monthly outputs were retained until 2030, from then until 2100 the monthly values were converted into annual averages by summing the monthly values for each year and dividing by 365 to get recharge in m/d.

Recharge values calculated by LASCAM can be inserted directly in to the model using the menu ***Props > Import > Database...***. LASCAM output files have the following format, one header line, followed by one line with the number of zones defined in the MODFLOW model and one line per zone including the recharge zone number and respective recharge value (in m/day). In case of the transient run, the number of zones and value lines are present for every stress period of the simulation. An illustration of the format of the recharge file generated by LASCAM is provided below.

```
Recharge Conc Ponding_Depth*All 47035
200
1 0.0000548 0 0
2 0.0000041 0 0
3 0.00001233 0 0
4 0.00001233 0 0
5 0.00001233 0 0
6 0.00001233 0 0
7 0.00001233 0 0
8 0.00001233 0 0
9 0.00001233 0 0
10 0.00001233 0 0
11 0.00001233 0 0
12 0.00001233 0 0
13 0.00001233 0 0
14 0.00001233 0 0
15 0.00001233 0 0
16 0.00001233 0 0
17 0.00001233 0 0
18 0.00001233 0 0
19 0.00001233 0 0
20 0.00001233 0 0
21 0.00001233 0 0
22 0.00001233 0 0
23 0.00001233 0 0
```

The LASCAM output can be converted into the correct format for import to MODFLOW using the 'Recharge_adjuster' EXCEL file. The following steps are required:

1. Update columns A to D in the worksheet 'modflow_recharge' with the new LASCAM recharge *.dat file
2. If required update the sub-catchment area sizes in Column AX in 'modflow_recharge' worksheet
3. Refresh the pivot table on 'pivot' worksheet
4. Copy the adjusted data from Columns Y and Z in 'modflow_recharge' worksheet to a new csv file

Recharge values of a given stress period can be copied to remaining periods in case constant recharge values are to be used in the model (not the case of Namoi model). The menu ***Props > Property Values > Copy Transient Data...*** opens a dialog which allows the operation to be conducted for one specific zone or all zones simultaneously.

2.10.2 *Mine inflow (drain boundaries)*

Drain boundary conditions can be edited in Groundwater Vistas using the boundary condition editing mode clicking the  button and selecting **Drain** in the combo box next to it. The editing mode can also be accessed through the menu ***BCs > Drain.***. The buttons for editing are the same as presented in the property mode.

The drains must be assigned for every stress period of the simulation, either by assigning and copying to the following periods, or assigning each stress period separately.

The data used to define the drain boundary conditions used to simulate groundwater flows to both open cut and underground mines in Groundwater Vistas in the predictive scenarios is generated using the spreadsheet "Namoi_CSG_Inputs_140512.xlsb".

Data needs to be inputted or modified in several of the tabs in the spreadsheet to produce and control the mine drain boundary condition file. The tabs relate either to drain cell specifics (conductance, stress periods etc) or to the model layer surfaces. The tabs are:

Drain boundary condition data:

- Drain cell conductance (Conductance tab)
- Groundwater Vista row and column coordinates of the mine drain cells, list of the mines including their type and the targeted seam (Hypothetical_Mines_Data tab)
- The model stress periods (SP tab)

Layer surface data

- Groundwater Vista export file of layer 12 (Hoskissons seam) top elevation (Elevations_L12 tab).
- Groundwater Vista export file of layer 14 (Melville seam) top elevation (Elevations_L14 tab).
- Groundwater Vista export file of layer 17 (Maules Creek formation) top elevation (Elevations_L18 tab).

How the spreadsheet works

The file includes five working tabs based on the seam targeted and the mine type: Hoskissons open cut mine, Hoskissons underground mine, Maules Creek open cut mine, Maules Creek underground mine and Melville open cut. In the 6 Scenarios there were no Melville underground mines.

The user chooses in the list of mine drain cells a particular seam target and mine type and then copy and pastes the data (mine numbers, row and column coordinates) to the appropriate tab. The data relative to the different mine will then be assigned a stress period and head corresponding to the top elevation of the cells. For an open cut mine, all the mine drain cells are active from the first layer to the targeted seam layer since the beginning of the production. For an underground mine, six stages (each of 5 years duration) of production have been assumed. The underground mine drain cells are only active in the targeted seam layer.

How to use the spreadsheet

Set the conductance values for the mine drain cells in the "Conductance" tab. Set the start and end times of the mines in columns B to D in the "SP" tab. Only the dates are required as the lookup functions determine the stress periods for use in the Groundwater Vistas input file. Once these tasks are complete the final, and most involved task, is to define which mines will target which coal seams / formation and the time variant development of the underground mines. This is done by the following method (an example of a Hoskissons Seam open cut and underground mine is given, but the process is identical for Melville Seam and Maules Creek Formation mines).

1. Open cut mines

    - Hypothetical_Mines_Data tab: Filter columns A to C for the mine number in column A. Chose which mine numbers to filter based on the data in columns F to H. In the template spreadsheet provided mine numbers 10, 11, 15, 18, 20, 22, 23, 25 and 28 are open cut and within the Hoskissons Seam. Therefore to define the open cut Hoskissons Seam mines filter for these mine numbers. Copy the filtered results.

    - Hoskissons_OC tab: Paste the mine numbers, row and column coordinates (from above) to column A, B and C (zone in blue). If additional mine cells are included, the calculations need to be extended to further rows. Column J corresponds to the input data for the groundwater numerical model.

2. Underground mines

    - Hypothetical_Mines_Data tab: Filter columns A to C for the mine number in column A. Chose which mine numbers to filter based on the data in columns F to H. In the template spreadsheet provided mine numbers 13, 16, 21, 24, 29 and 31 to 34 are underground mines targeting the Hoskissons Seam. Therefore to define the underground Hoskissons Seam mines filter for these mine numbers. Copy the filtered results.

    - Hoskissons_UG tab: Paste the mine numbers, row and column coordinates (from above) to column A, B and C (zone in blue). If additional mine cells are included, the calculations need to be extended to further rows. Column J corresponds to the input data for the groundwater numerical model. Column D is used to define the development schedule of each underground mine. The development is split into 6 stages of 5 years each, and the active cells during each of these stages is defined by putting the stage number next to each mine cell in this column. This is done manually.

When finished, data for the Groundwater Vistas input file is generated in the "Import" tab. This must be copied and pasted into another blank spreadsheet which is then saved as a *.csv. This file can then be imported to Groundwater Vistas.

The text file (*.csv) contains eight columns (see figure below) described as follows:

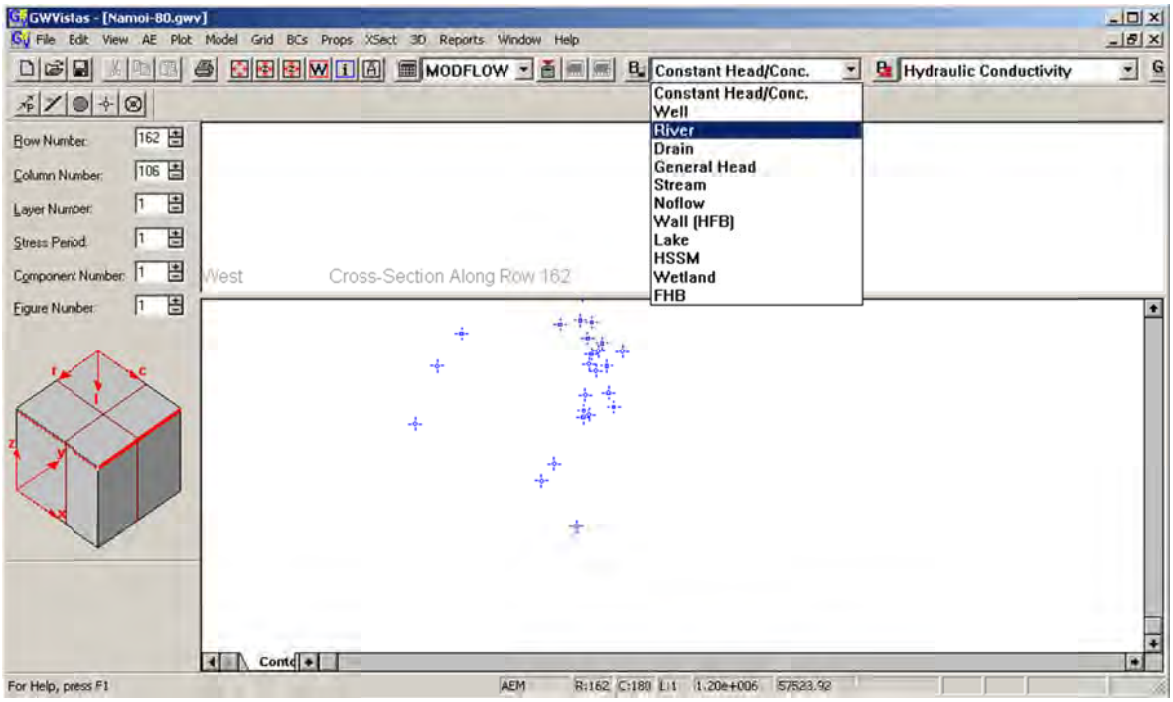
- First column (**R**) – Indicates the row index of the cell where the constant head will be applied
- Second column (**C**) – Indicates the column index of the cell where the constant head will be applied
- Third column (**L**) – Indicates the layer index of the cell where the constant head will be applied
- Fourth column (**Head**) – Indicates the head elevation for the constant head to be applied (in mAHD for the Namoi model)
- Fifth column (**Reach**) – Reach index used by Groundwater Vistas, and used here to index the different mines
- Sixth column (**S**) – Indicates the starting stress period for the constant head
- Seventh column (**E**) – Indicates the ending stress period for the constant head
- Eighth column (**Cond**) – Indicates the boundary conductance (not relevant for constant heads).

R	C	L	Head	Reach	S	E	Cond
265	143	5	268	31	1	18	50
265	143	4	268	31	1	18	50
265	143	3	268	31	1	18	50
265	143	2	268	31	1	18	50
265	143	5	268	31	19	30	50
265	143	4	268	31	19	30	50
265	143	3	268	31	19	30	50
265	143	2	268	31	19	30	50
265	143	2	268	31	31	42	50
265	143	3	268	31	31	42	50

The text file can be imported using the menu ***BCs > Import > Text File...*** .

2.10.3 *Rivers / streams*

Similar to constant heads, the river boundary conditions can be accessed entering in the boundary condition editing mode using the  button and selecting **River** in the combo box, as illustrated below:

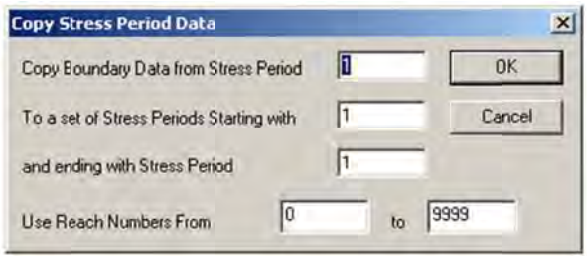


The editing mode can also be accessed using the menu **BCs > River**. Editing options and toolbar buttons are the same presented in the property mode.

River boundary conditions must be assigned for all stress periods of the simulation. This can be achieved editing one stress period at a time, or if this boundary does not change throughout the simulation, assigning one stress period and copying these settings to the remaining ones.

For editing one stress period at a time, the stress period to be set can be accessed using the **Stress Period** text box located to the left tool bar. Typing a number in the text box will lead to the corresponding stress period.

Settings from one stress period can be copied to another using the menu ***BCs > Modify > Copy Stress Period...*** which opens the following dialog:



The first textbox (**Copy Boundary Data from Stress Period**) sets the stress period which settings will be copied. The next two text boxes define the initial and final stress periods to which the settings will be copied.

The two text boxes in **Use Reach Number From** define which boundary condition reaches will be copied from one stress period to the others.

Complex definition of river boundary may be required at times, especially when highly spatial and temporal variability is required. Similar to constant heads described previously, the river boundary conditions can be edited outside Groundwater Vistas and once finished can be imported into the model. This was the method adopted for the Namoi model.

The file must obey the river package format defined for USGS MODFLOW 2000. The format is barfly described below:

- First line contains the number equating to the number of lines in the file;
- For each stress period, one header line containing the number of record lines for the stress period, followed for one line for each record;
- For each record, six columns are defined as follows:
    - Column 1 – Row index;
    - Column 2 – Column index;
    - Column 3 – Layer index;
    - Column 4 – River head (in mAHD for the Namoi model);
    - Column 5 – Boundary conductance;
    - Column 6 – River bottom elevation;
    - Column 7 – Auxiliary variables not applicable to Namoi Model.

Once the file is prepared it can be imported using the menu **BCs > Import > MODFLOW Package...**. In the combo box **Files or type:**, located at the bottom of the dialog, choose the river option and select the file to be imported accordingly.

The generation of river boundary conditions for a model of this size and this many stress periods is a complex task. It has required the use of several macros and large complex spreadsheets used to interpolate between river gauges and between missing temporal data. Now that the process of building these inputs has been completed the most efficient method of making any changes to these inputs will be via the Groundwater Vistas interface, using the methods described above. Single river cell settings (stage, river bottom elevation and conductance) can be modified in this way or entire reaches.

2.10.4 *Abstraction wells (background)*

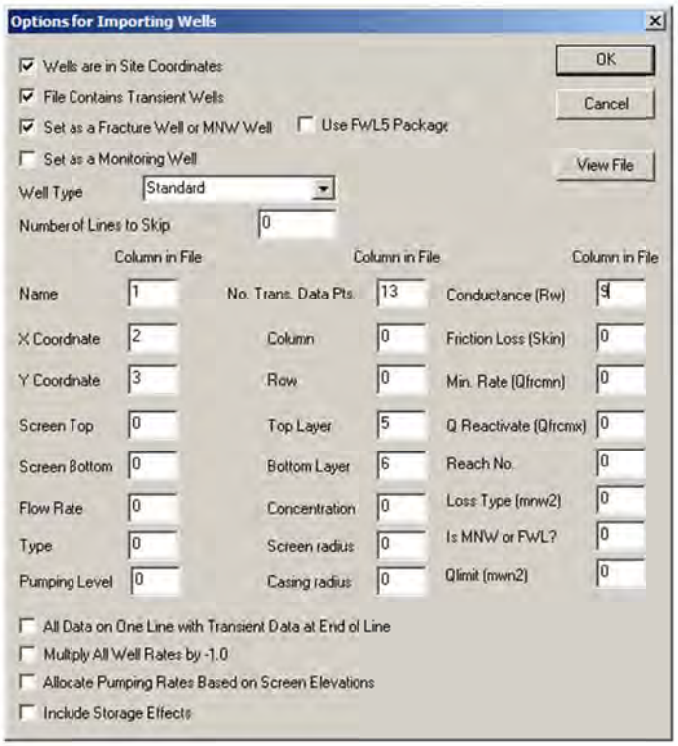
The background abstractions (irrigation, public water supply etc) have been simulated in the Groundwater Model as MNW wells. The transient data has been compiled into a comma delimited file (.csv). Each abstraction point has a header line followed by a rate (m^3/d) for each of the 304 stress periods, even if that rate is zero. The following architecture is used:

- Well header line (values assigned to columns not mentioned below are not used in the file import and are therefore not important):
    - Column 1 – Well name
    - Column 2 – Well easting
    - Column 3 – Well northing
    - Column 5 – Top layer of well
    - Column 6 – Bottom layer of well
    - Column 9 – Well Rw value

- Column 13 – number of transient data points

- Abstraction rate lines
  - Column 1 – Starting stress period
  - Column 2 –Ending stress period
  - Column 3 – Rate (m^3/d)
  - Column 4 – Concentration (default 0)

Once the file is prepared it can be imported using the menu **AE > Import > Well text file...**. which opens the following dialog:



The dialog should be filled in as shown above. The option **Set as a Fracture Well or MNW well** should be selected and the column numbers filled in for **Name, X coordinate, Y coordinate, No. Trans. Data Pts, Top Layer, Bottom Layer** and **Conductance (Rw).**

2.10.5 *Abstraction and injection wells (CSG)*

Abstraction from the CSG wells is simulated using the WEL package. Injection of treated water back into the model domain is simulated in the same way. The transient data has been compiled into a comma delimited text file (.csv) with the same characteristics as that described above for the background abstractions.

The input file is generated using the spreadsheet "Namoi_CSG_Inputs_140512.xlsb".

User defined inputs to the spreadsheet relate to one of three types; layer surfaces, CSG field geometry / abstraction and stress period set-up. These are described below:

Layer surface inputs

- Groundwater Vista grid coordinates of the cells where only Hoskissons seam is observed (GV_points_Hoskissons tab)
- Groundwater Vista grid coordinates of the cells where only Melville seam is observed (GV_points_Melville tab)
- Groundwater Vista grid coordinates of the cells where only Maules Creek formation is observed (GV_points_Maules tab)
- Groundwater Vista grid coordinates of the cells where both Hoskissons and Melville seam are observed (GV_points_Hoskissonsmelville tab)
- Groundwater Vista grid coordinates of the cells where both Hoskissons seam and Maules Creek formation are observed (GV_points_Hoskissonsmaules tab)
- Groundwater Vista grid coordinates of the cells where both Melville seam and Maules Creek formation are observed (GV_points_Maulesmelville tab)
- Groundwater Vista export file of layer 12 (Hoskissons seam) thickness (Elevations_L12 tab).
- Groundwater Vista export file of layer 14 (Melville seam) thickness (Elevations_L14 tab).
- Groundwater Vista export file of layer 18 (Maules Creek formation) thickness (Elevations_L18 tab).

CSG inputs

- Groundwater Vista grid coordinates of the cells within the coal seam gas project area. (CSG_Field_Pts tab)
- The yearly average field production or injection rates (QC_Inputs tab)

Stress period inputs

- The model stress periods (SP tab)

How the spreadsheet works

Every model cell within a CSG field is defined as being one of the following based on the thickness of the coal seams or formations they intercept:

- "Hoskissons". Only the Hoskissons Seam is present at a thickness of 5 m or greater in this cell
- "Melville". Only the Melville Seam is present at a thickness of 5 m or greater in this cell
- "Maules Creek". Only the Maules Creek Formation is present at a thickness of 5 m or greater in this cell
- "Hoskissons-Melville". Both the Hoskissons and Melville Seams are present in this cell, both at a thickness of 5 m or greater
- "Hoskissons-Maules". Both the Hoskissons Seam and Maules Creek Formation are present in this cell, both at a thickness of 5 m or greater

- "Melville-Maules". Both the Melville Seam and Maules Creek Formation are present in this cell, both at a thickness of 5 m or greater

Cells where all three seams / formations were present are limited and were therefore discounted from this analysis. There is therefore no "Hoskissons- Melville-Maules" class.

If there is no thickness of coal seam or formation in any particular model cell then it is not assigned to any of these groups. This data is used directly to assign wells in a CSG field to the correct layer and to apportion the abstraction accordingly.

A single well can only target one of the three geological units. Therefore a cell where two or more coal seams / formations are present will have two or three wells assigned.

If a cell contains only one well, the well production (or injection) rate corresponds to the cell production (or injection) rate. If a cell contains two wells, the well production (or injection) rates are calculated based on the cell production (or injection) rates proportionally to the targeted unit thickness.

The calculations described above are undertaken in the Field_Cells_Calc, Bando_Hoskissons, Bando_Melville, Bando_Maules tabs and the results are amalgamated in the CSG_Field_GV_Input tab.

How to use the spreadsheet

In order to produce a full CSG abstraction input file for Groundwater Vistas the following tabs require inputs.

1. QC_Inputs tab: Input the yearly average field production (or injection) rates in m^3/d in columns A and B. Input positive values for a production project (abstraction) and negative values for injection project.

2. CSG_Field_Pts tab: Input cell coordinates (from Groundwater Vista model grid – X, Y, row and column – highlighted in yellow) of all model cells falling within the CSG field area.

3. Field_Cells_Calc tab: Check if all the CSG field cells (see step 2) are taken into account and if they are not extend row 1822 down (which holds the calculations)

4. Hoskissons tab: In Field_Cells_Calc tab select all cells beneath and including row 5, columns A to P and filter column C for the definitions "Hoskissons", "Hoskissons-Maules" and "Hoskissons-Melville". Copy and transpose paste the following:

    a. Column M to cell G3 in the Bando_Hoskissons tab

    b. Column N to cell G4 in the Bando_Hoskissons tab

    c. Column J to cell G5 in the Bando_Hoskissons tab

5. Melville tab: In Field_Cells_Calc tab select all cells beneath and including row 5, columns A to P and filter column C for the definitions "Hoskissons", "Hoskissons-Maules" and "Hoskissons-Melville". Copy and transpose paste the following:

    a. Column M to cell G3 in the Bando_Melville tab

    b. Column N to cell G4 in the Bando_Melville tab

    c. Column J to cell G5 in the Bando_Melville tab

6. Maules tab: In Field_Cells_Calc tab select all cells beneath and including row 5, columns A to P and filter column C for the definitions "Hoskissons", "Hoskissons-Maules" and "Hoskissons-Melville". Copy and transpose paste the following:

    a. Column M to cell G3 in the Bando_Maules tab

    b. Column N to cell G4 in the Bando_Maules tab

    c. Column J to cell G5 in the Bando_Maules tab

7. QC_Inputs tab: Quality check the results. Column D should be equalled to 0% as this is where the input (and desired) total yearly average abstraction for the entire CSG field is compared against the final product of the spreadsheet calculations.

8. CSG_Field_GV_Input**:** If more wells than currently in the file are considered, extend columns F, P and Z. Copy and paste columns G, Q and AA (one under the other) to a *.csv file. This is the final input file for the CSG abstractions for Groundwater Vistas.

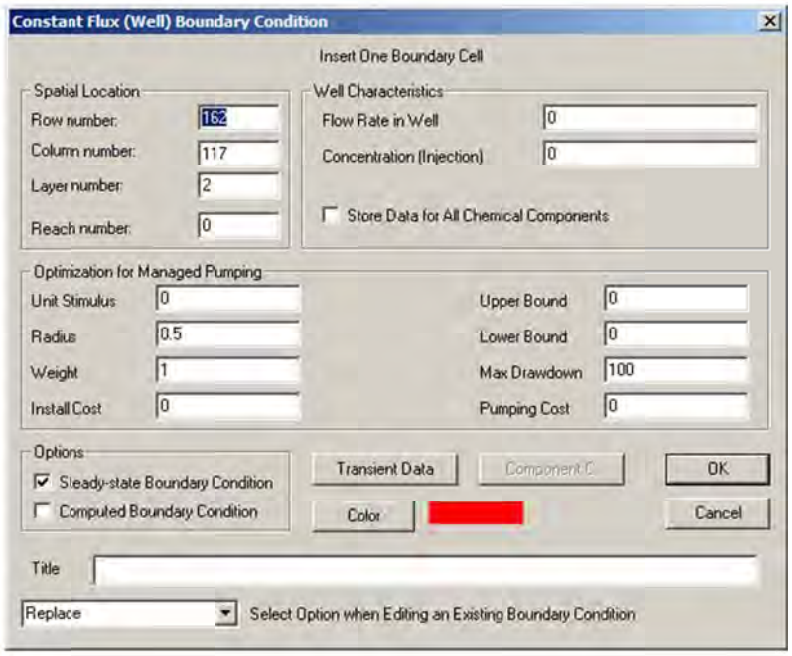
If additional wells are included, check in the tabs above that all the data are taken into account in the calculations. If not, the calculations need to be extended to further rows or columns.

Once the file is prepared it can be imported using the menu **AE > Import > Well text fie...** which opens the dialog displayed above. On this occasion, as MNW wells are not to be used, leave the option **Set as a Fracture Well or MNW well** unselected and specifying a column number for **Conductance (Rw)** is not required. These wells are therefore imported as analytical elements, but when datasets are created the data is passed into a WEL package file, rather than an MNW package file.

2.10.6 *Irrigation (injection) wells*

Groundwater recharge from irrigation is simulated using the WEL package. This boundary condition was set up directly in Groundwater Vistas using the process described below.

Activation of the WEL editing menus is accomplished by using the menu **BCs > Well...**. Once this has been selected wells can be assigned directly to model cells by selecting the relevant layer (for irrigation recharge this was always Layer 1), locating the mouse arrow over the desired cell and pressing the right mouse button. This action opens the following dialog:



As the irrigation recharge varies from month to month the input cannot be treated as steady state. For this reason the option **Steady-state Boundary Condition** should be unselected. Once this has been done time variant rates of recharge can be assigned by left clicking on the **Transient Data** option.

The spreadsheet "Namoi_Transient_Irrigation.xlsx" has been set up to allow production of both time variant historical (based on 306 equal length stress periods) and time variant predictive (based on 304 unequal length stress periods) input files. To change the irrigation rates on which the inputs are calculated change the values in cells H2 (currently equal to 30 mm/yr) or cell I2 (currently equal to 70 mm/yr). Recalculate by pressing F9 and the data required as input to the Groundwater Vistas boundary conditions is displayed in columns T to V (30 mm/yr historical), X to Z (70 mm/yr historical), AD to AF (30 mm/yr predictive) and AH to AJ (70 mm/yr predictive).

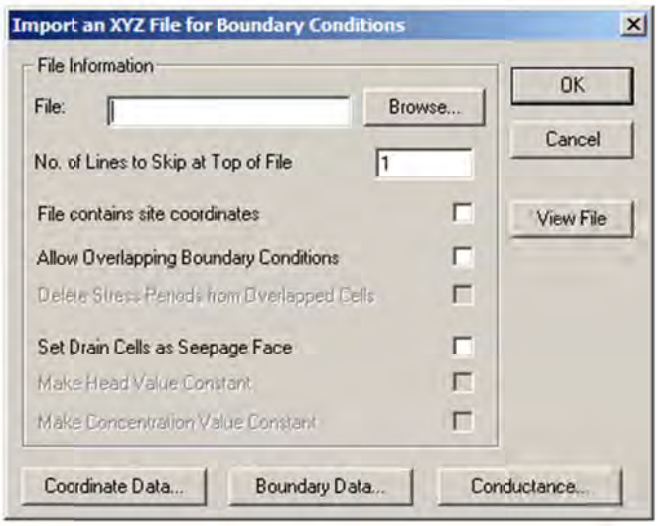
The data within these columns can then be copied and pasted directly into the **Transient Data** dialog described above.

The process described above was undertaken once for the historical model and once for the predictive model. To allow this input to be used in the creation of other scenarios and sensitivities, it was exported from Groundwater Vistas as a text file. The text file has the following attributes:

- Four header lines. The first 2 correspond to wells and the second two correspond to rivers. As the irrigation recharge is simulated as wells it is the first two lines that are relevant here. Followed by;
- Well data lines. For each stress period a rate is supplied for each irrigation recharge well. The data is organised as follows (only those options used in the import process are described):
    - Column 1 – Row
    - Column 2 – Column
    - Column 3 - Layer
    - Column 5 - Rate
    - Column 7 – Starting stress period

- Column 8 – Ending stress period

The text file can be imported by first activating the well boundary condition options (**BCs > Well...**). Then, using the menu **BCs > Import > Text file...** which opens the following dialog box:



Using this dialog box browse to the location of the input file. The lines to skip at the top of the file should be set to 4. No other changes are required, but the options at the bottom (**Coordinate Data** and **Boundary Data**) must be entered and filled out so that the import columns match the text file. For example, once the **Boundary Data** option has been selected the following dialog box will open and should be filled out as indicated:

2.10.7 *Connective cracking / permeability enhancement*

Changes to model parameters in order to simulate connective cracking above underground mines has been undertaken manually. Shapefiles were created delineating zones where changes were required. In order to do this the shapefiles need to be imported as maps. Once the file is prepared it can be imported using the menu **File > Map > Shapefile...**. The file must then be selected. Groundwater Vistas will then use this file to produce a file in its own format (.map). A name for this file must be provided.

Once the maps are imported and can be seen in Groundwater Vistas the hydraulic parameters can be changed by first selecting **Props > Hydraulic Conductivity....** This activates the options for controlling this parameter. If changes to storage are required **Storage/Porosity** must be selected instead. Once this has been done changes to the parameters can be made simply by right clicking on relevant model cells. To control the zone number attributed to the cell when this is done, simply select **Props > Default Values...**. Then, in the dialog box that opens, set the **Default Zone Number** to equal the zone that is required.

2.11 Model and translation and run

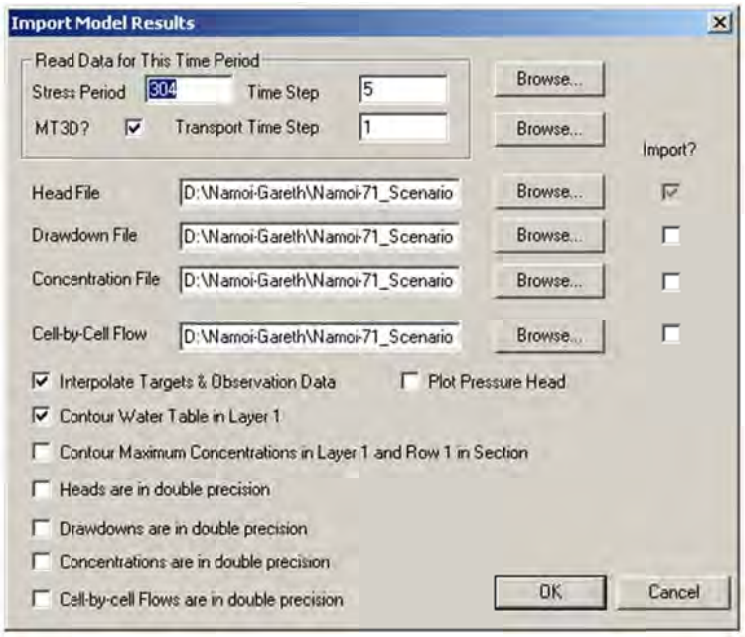
Once model setup is finished, the model inputs need to be translated to MODFLOW input files format. This operation is done pressing the  button or through the menu ***Model > MODFLOW > Create Datasets***. With the model files translated, the model can be set to run wither using Groundwater Vistas or with native USGS MODFLOW through the command prompt.

From Groundwater Vistas, the model can be run using the menu ***Model > MODFLOW > Run MODFLOW***. A new dialog will appear showing the running progress and verbose messages. The model should carry on running through the time steps and stress periods until it gets to the end. At this point it will say the Modflow has finished and model results will be ready to be post-processed.

2.12 Post processing results

2.12.1 *Importing model results*

Prior to any post-processing the model results generated by MODFLOW have to be imported into Groundwater Vistas. Model results can be imported using the button  or using the menu ***Plot > Import Results...*** . A new dialog will appear and it is illustrated in the following figure.



Type the numbers corresponding to the desired stress period and time step in the text boxes located in the box **Read Data for This Time Period** and press the **OK** button. By default, Groundwater Vistas will import hydraulic head results, however, drawdown and cell-by-cell flows can also be loaded marking the **Import** checkboxes next to the **Drawdown File** and **Cell-by-Cell Flow** text boxes.

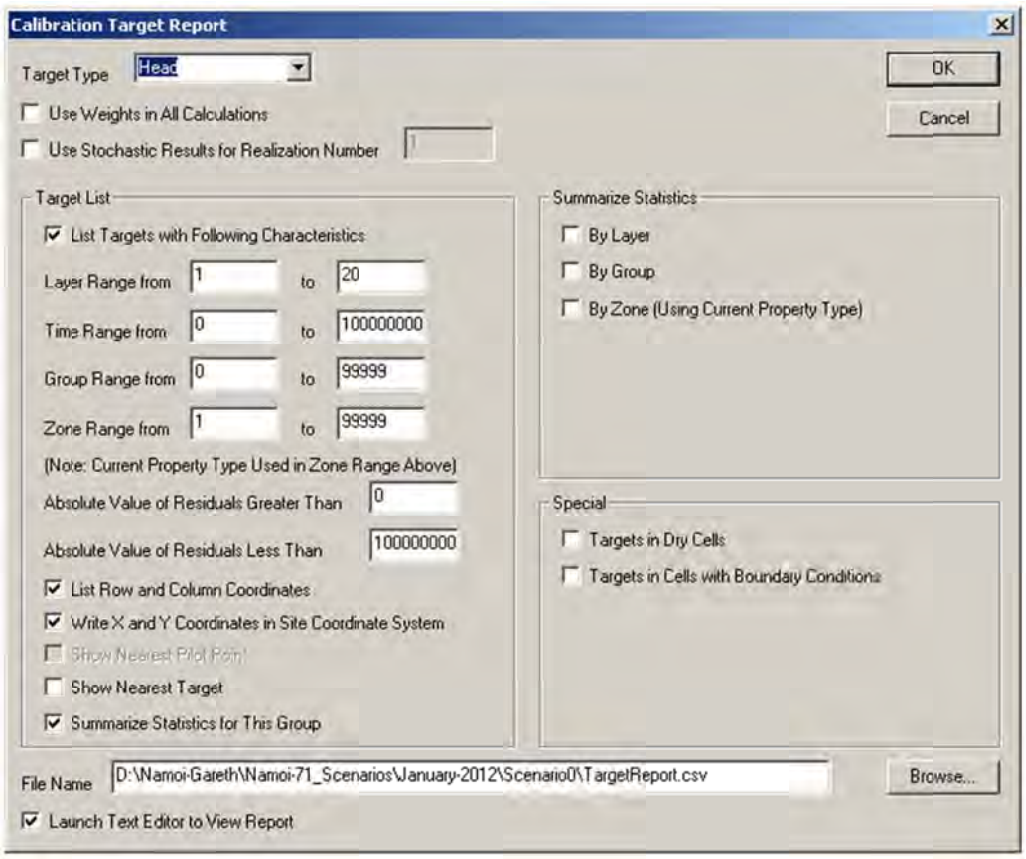
Important: Only results specified in the output control are saved by MODFLOW and, therefore, only these results can be imported into Groundwater Vistas.

2.12.2 *Contours*

For the most part model results are reported as drawdown from coal and gas developments. As the model predicts drawdown from all sorts of things on top of this (background abstraction, natural recharge changes etc) the results must be compared against Scenario 0. Drawdown contours are produced by subtracting Surfer ascii grids from the Scenario 0 model from Surfer ascii grids from the Scenario 3 model. If the grids are exported from the same stress period and time step, this calculation provides the drawdown only due to coal and gas developments at that time. To export the grids the head results from the stress period and time step required must be imported following the process outlined above. Then selecting the **Plot > What to Display...** dialog box the option **Display Contours of Head** must be selected. Once this has been done head grids can be exported for the active layer by selecting **File > Export...**. In the dialog box that appears Shapefiles and many other formats can be selected as the **Save as type**.

2.12.3 *Hydrographs*

Hydrographs for the observation targets can be exported using the menu ***Reports > Calibration > Target Residuals...*** . The **Calibration Target Report** dialog will appear as illustrated in the next figure.



Targets to be exported can be filtered by layer, simulation time, group and zone using the **Target List** box. Summarized statistics by layers, group and zone can be generated by ticking the checkboxes located in the **Summarize Statistics box**.

The location and name of the CSV file containing the hydrographs is defined in the **File Name** text box located at the bottom of the dialog. Checking the option **Launch Text Editor to View Report** will automatically open the report once it is generated. To generate report, press the **OK** button located in the top right corner.

Targets can be defined manually in Groundwater Vistas using the  button or the menu ***AE > Target***. Alternatively, targets can be imported using the menu ***AE > Import > Target from Text File...***

The format used for the targets is described in Groundwater Vistas documentation and consists basically of a CSV file containing the following:

- One initial head line with column identifiers;
- For each target :
    - One line with the target information including name, coordinates, row, column and layer indexes, etc...
    - One line for every observation containing time, observed value and weight (irrelevant). Dummy observation values can be used to generate the target hydrographs.

The following figure illustrates the format used by the target text files:

Name	X	Y	Target	Layer	row	column	Weight	Group	Layer2	Min_K	Max_K	Compone	Type	Layer_Elevation	NumTimes
Zone11-Layer1	794051.6	6625656	0	1	148	151	1	1	0	0	0	1	Head	247.419991	303
62	250	1													
92	250	1													
123	250	1													
153	250	1													
184	250	1													
215	250	1													
243	250	1													
274	250	1													
304	250	1													
335	250	1													
365	250	1													
396	250	1													
427	250	1													
457	250	1													
488	250	1													
518	250	1													
549	250	1													

Calibration spreadsheets

Two spreadsheets are used to compare the predicted groundwater levels from the historical model against the observed groundwater levels. The first "Namoi_Modflow_Calibration_General.xlsb" includes the public data that was used to assess the calibration in the Upper Namoi Alluvium. The second "Namoi_Modflow_Calibration_Mines.xlsb" includes monitoring associated with mining projects. Both are described in more detail below:

Namoi_Modflow_Calibration_General.xlsb

The target file "Target_Calibration_General_08092011.csv" must first be imported into Groundwater Vistas so that predicted groundwater levels through time can be exported at the correct locations from the model. This process is described above.

To update the calibration hydrographs paste all of the Groundwater Vistas exported data (including the header line) into cell A6 of the tab "GWVOutput". Then press F9 to recalculate and the graphs located in the other tabs will update.

Namoi_Modflow_Calibration_Mines.xlsb

The process is the same for the analysis of the model predictions against mine related groundwater levels. The target data file is called "Target_Calibration_mines__241011.csv". The exported groundwater levels from these targets can be pasted into cell A2 of the tab "GWVOutput. Then press F9 to recalculate and the graphs located in the other tabs will update. The hydrographs used in the calibration are found in red coloured tabs ("Narrabri", "Rocglen", "Sunnyside" and "Boggabri"). Other hydrographs that were available are found in the uncoloured tabs.

Prediction hydrograph - alluvial

As with the calibration hydrographs, the initial step in reproducing the hydrographs of simulated drawdown at the hypothetical alluvial monitoring locations is to import the targets ("Namoi_Hypothetical_Targets_Alluvium.csv") into Groundwater Vistas. Once the standard procedure of importing the targets and model results has been completed the results can be exported at these locations and the results imported into columns B to K of the tab "ScX_Results" found in the file "Namoi_Predictive_Hypothetical_Alluvium_Hydrographs.xlsb". If background settings (recharge, background abstraction, irrigation etc, or any more fundamental model changes made) then the updated Scenario 0 model results must also be exported in this format and pasted into columns B to K of the tab "Sc0_Results". The difference between the Scenario 0 and Mining / CSG scenario run (the drawdown due to coal and / or CSG development) is then displayed in the hydrographs in the tab "Plots".

Prediction hydrographs - hard rock (HR) hydrographs

The file containing the HR target locations and data is called "Namoi_Hypothetical_Targets_HR.csv". A single spreadsheet is devoted to each hypothetical location due to the size of the output data. These spreadsheets are called:

- Namoi_Predictive_Hypothetical_Hydrographs-Bando1.xlsx
- Namoi_Predictive_Hypothetical_Hydrographs-Bando2.xlsx
- Namoi_Predictive_Hypothetical_Hydrographs-Bando3.xlsx
- Namoi_Predictive_Hypothetical_Hydrographs-Bando4.xlsx
- Namoi_Predictive_Hypothetical_Hydrographs-Narrabri1.xlsx
- Namoi_Predictive_Hypothetical_Hydrographs-Narrabri2.xlsx
- Namoi_Predictive_Hypothetical_Hydrographs-Narrabri3.xlsx
- Namoi_Predictive_Hypothetical_Hydrographs-Narrabri4.xlsx

The configuration of each spreadsheet is the same and the exported data must be pasted into columns K to T of the "Results" tab. If Scenario 0 has also changed, this data can be pasted into columns A to J. The difference between Scenario 0 and the coal and gas development scenario is then calculated in column U and the hydrograph presented in the tab "Plot".

2.12.4 *Historical surface water / groundwater interaction*

The difference between Scenario 0 and the coal and gas development scenario predictions of interaction flows between groundwater and surface water are analysed using three spreadsheets:

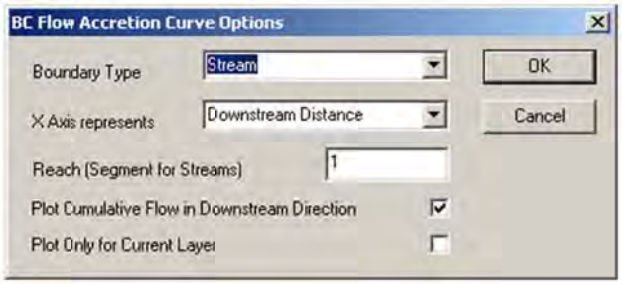
- Namoi_river_Accretion-2030.xlsx
- Namoi_river_Accretion-2030.xlsx
- Namoi_river_Accretion-2030.xlsx

Following the method above, the predictive data can be pasted directly into the relevant column in each "Reach" tab in the spreadsheet (there are 14 of these tabs, one for each modelled reach). Results for any coal and gas scenario can be pasted into cell B3 and a re-run scenario 0 into cell C3.

The results are then displayed in the hydrographs in the "Plots" tab.

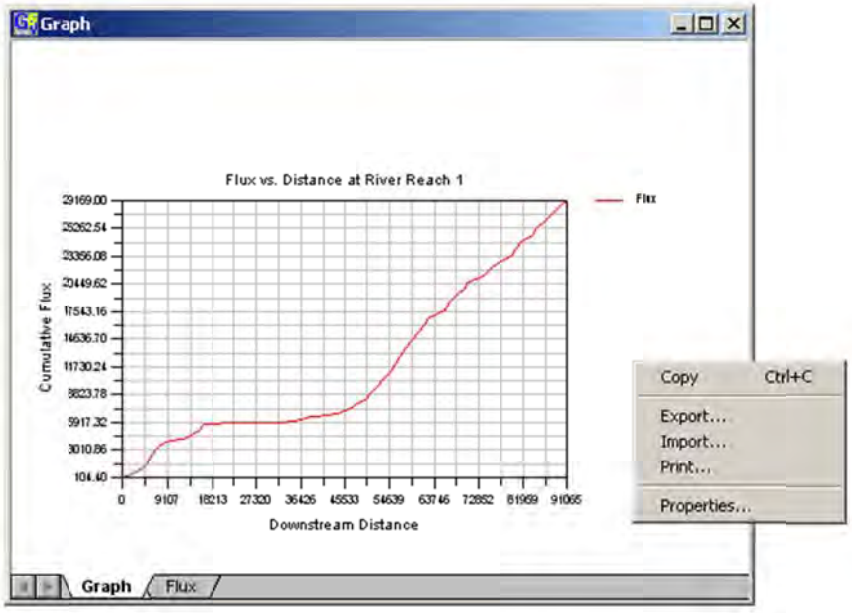
Results from surface/water groundwater flow exchanges can be extracted using the menu ***Plot > Import Results...*** as described in section 2.11.1. In this case the check box for importing **cell-by-cell flow** must be marked.

With the results imported, the calculations can be made through the menu ***Plot > Mass Balance > BC Flow Accretion Curve...*** The dialog in illustrated in the next figure appears.



The combo box **Boundary Type** allows the selection of boundary type used for the calculations. For surface water/ groundwater interaction, the option **River** must be chosen. Select the option **Downstream Distance** in the **X Axis represents** combo box and type the desired river reach ID in the text box **Reach (Segment for Streams)**. The reach id is defined during the setup of the river boundary conditions described in section 2.9.3.

With all options selected pressing the **OK** button displays a chart window with a plot of flux rates against downstream distance. To copy the numeric values, right-click on the graph and select the **Copy** option. The data will be sent to Windows clipboard and can be pasted as a spreadsheet in Excel or as text.



2.12.5 *Predictive period mass balance*

The Groundwater Vistas files are set up with hydrostratigraphic property zones for use with mass balance analysis. The zones are:

- Zone 1: Lower Namoi Alluvium (layer 2)
- Zone 2: Upper Namoi Alluvium – Narrabri Formation (layer 1)
- Zone 3: Upper Namoi Alluvium – Gunnedah Formation (layer 2)
- Zone 4: Unused
- Zone 5: Hard rock formations (layers 2 to 20)

Once the cell by cell flow data (the *.cbb file) has been imported to Groundwater Vistas the flows calculated at boundary conditions within these zones and flows between neighbouring zones can be exported.

The mass balance data can be exported by using the menu **Plot > Mass Balance > HydroStratigraphic Units > Export HSU Report...** which opens a simple dialog box. A file name must be chosen (*.csv) and then when prompted "Summarize Mass Balance for All Times" select "yes".

The exported data can then be pasted directly into the Sc0 or ScX tabs of the analysis spreadsheets depending on requirements. The results are displayed in the tab "Plot". The process is the same for mass balance, groundwater / surface water interaction and flow spreadsheets, which are detailed below:

- Namoi_Mass_Balance.xlsx
- Namoi_Interaction.xlsx
- Namoi_flows.xlsx

THIS PAGE HAS BEEN LEFT BLANK INTENTIONALLY

3 HYDROLOGIC MODEL

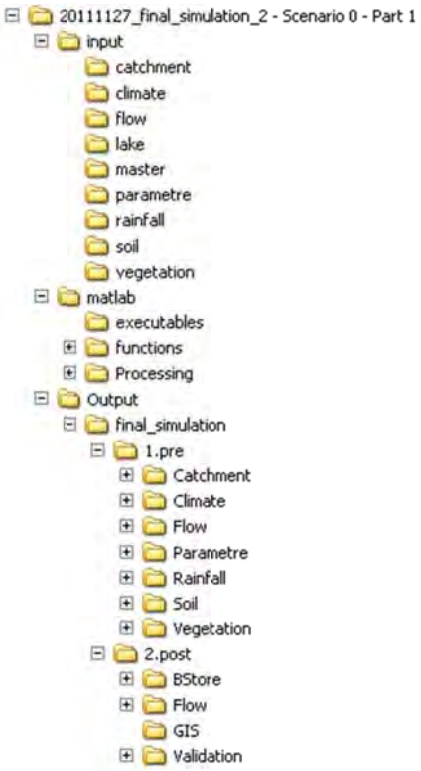
3.1 Introduction

The Large Scale Catchment Model (LASCAM) developed by Viney and Sivalapan (2000) was the selected code to simulate the surface water processes occurring in the catchment. While providing a robust simulation of processes such as infiltration and run-off with a relatively small number of parameters, LASCAM does not have a graphical user interface for pre and post processing of model inputs and results.

In order to allow the use of LASCAM in an optimized way and accelerate workflows related to model settings and results processing, additional scripts were developed using Matlab. Matlab is a high-level language and interactive environment for technical computing developed by Mathworks. The Matlab scripts help translating the inputs stored in spreadsheets into the LASCAM input files, as well as processing LASCAM results and generating ready-to-use outputs such as charts, spreadsheets and MODFLOW package files (recharge in this case).

The input files, Matlab scripts and LASCAM executables were built in an enclosed folder structure as illustrated below. The structure contains 3 main folders, namely **input**, **matlab** and **output**. The **input** folder contains the inputs created by the user. The folder **matlab** contains matlab scripts, the input files translated to LASCAM format (once the input files are translated) and LASCAM executable, while the folder **output** contains the post-processed LASCAM results.

Important: The folder structure must not be modified otherwise the Matlab scripts will not operate properly.



All the inputs of the LASCAM model are saved in the folder **Input**. The several inputs are divided in subfolders according to its nature, as follows:

- **Catchment** – this subfolder contains information about sub-catchments;
- **Climate** – this subfolder contains climate information, such as rainfall and evaporation;
- **Flow** – this folder contains flow observations used in the model calibration;
- **Lake** – this folder contains information about lakes and reservoirs;
- **Master** – this folder contains inputs that control running time and outputs;
- **Parameter** – this folder contains parameter inputs that are common to all sub-catchments;
- **Rainfall** – this folder contains rainfall data;
- **Soil** – this folder contains information regarding soil parameters; and
- **Vegetation** – this folder contains information on the vegetation and impervious surfaces.

The files in the folder **matlab** relate to internal operation of the models and therefore must not be modified in any way. Details on the **output** folder and post processing are presented in Section 3.6.

3.2 Domain and sub-catchment selection

The LASCAM model domain and sub-catchment selection is defined in the file **catchment.csv** located in the **Catchment** folder. The file consists of a header line containing the column identifiers followed by one line per sub-catchment in the model. The sub-catchment input consists of 9 parameters, namely:

- **Link** – indicates the sub-catchment ID, starting from 1 for the most downstream catchment up to the total number of catchments (99 in the Namoi LASCAM model);
- **Dslink** – indicates the ID of the downstream sub-catchment. For the most downstream sub-catchment, Dslink must be assigned as 0;
- **DistToOF** – distance along the river from the centroid of the sub-catchment to the most downstream point of the basin;
- **BasinArea** – total area of all sub-catchments located upstream of the current sub-catchment;
- **LinkArea** – area of the sub-catchment;
- **DrainDensity** – ratio between stream length within the sub-catchment and its corresponding area;
- **Y** – northern coordinate of the sub-catchment centroid;
- **X** – eastern coordinate of the sub-catchment centroid; and
- **Projection** – coordinate projection system used to define the centroid coordinates.

Important: Downstream sub-catchments must always have a lower index that those located upstream, and the most downstream sub-catchment must have index 1.

3.3 Parameterisation

3.3.1 *Soil settings*

Soil settings can be accessed and modified in the CSV file **soil.csv** in the **soil folder**. This file consists of one header line with column names followed by one line per sub-catchment where the parameters are set.

The required parameters are:

- **Link** – corresponds to the sub-catchment ID;
- **Dmin** – minimum soil depth (m);
- **Dmean** – average soil depth (m);
- **Poroup** – top-soil porosity (-);
- **FieldCap** – top-soil field capacity (-);
- **PoroZNS** – deep soil porosity (-);
- **DepthBR** – depth to bedrock (m);
- **Psif** – bubbling pressure (mm);

- **Lambda** – soil index;
- **SpecYield** – specific yield (-); and
- **AlphaGW** – fraction of the sub-catchment underlain by contributing aquifers.

The sub-catchment lines have to be in ascending order by ID. A typical soil file format is shown below.

link	dmin	dmean	poroup	fieldCap	poroZNS	depthBR(r	psif(mm)	lambda	specYield	alphaGW
1	0.2	2.1	0.313901	0.06805	0.213901	20	100	0.33	0.245851	0.01
2	0.2	2.1	0.289161	0.080419	0.189161	20	100	0.33	0.208742	0.01
3	0.2	2.1	0.322461	0.06377	0.222461	20	100	0.33	0.258691	0.01
4	0.2	2.1	0.295236	0.077382	0.195236	20	100	0.33	0.217854	0.01
5	0.2	2.1	0.338372	0.055814	0.238372	20	100	0.33	0.282557	0.01
6	0.2	2.1	0.272875	0.088563	0.172875	20	100	0.33	0.184312	0.01
7	0.2	2.1	0.303253	0.073374	0.203253	20	100	0.33	0.229879	0.01
8	0.2	2.1	0.28909	0.080455	0.18909	20	100	0.33	0.208634	0.01
9	0.2	2.1	0.315647	0.067177	0.215647	20	100	0.33	0.24847	0.01
10	0.2	2.1	0.3257	0.06215	0.2257	20	100	0.33	0.26355	0.01
11	0.2	2.1	0.268375	0.090812	0.168375	20	100	0.33	0.177563	0.01
12	0.2	2.1	0.254548	0.097726	0.154548	20	100	0.33	0.156822	0.01
13	0.2	2.1	0.262164	0.093918	0.162164	20	100	0.33	0.168246	0.01
14	0.2	2.1	0.282027	0.083986	0.182027	20	100	0.33	0.198041	0.01
15	0.2	2.1	0.278619	0.085691	0.178619	20	100	0.33	0.192928	0.01
16	0.2	2.1	0.251348	0.099326	0.151348	20	100	0.33	0.152022	0.01
17	0.2	2.1	0.260748	0.094626	0.160748	20	100	0.33	0.166122	0.01
18	0.2	2.1	0.324071	0.062964	0.224071	20	100	0.33	0.261107	0.01

3.3.2 *Surface water storage settings*

Parameters of surface water storage such as lakes and reservoirs are defined in the CSV file **lake.csv** located in the **lake folder**. Similar to the soil file, the lake file consists of one header line followed by one line per lake/storage structure. Required parameters are described as follows:

- **Sub** – Id of the sub-catchment where the lake/storage is located;
- **LakAMax** – Maximum area of storage (km^2);
- **LakvA** – Parameter relating storage volume and area;
- **LakVDead** – Dead volume of storage (ML);
- **LakVAMax** – Storage volume at LakAMax (ML);
- **LakVMax** – Maximum storage volume (ML);
- **LakQMax** – Maximum storage discharge;
- **LakvQ** – Parameter relating volume and downstream discharge (-);
- **NAME** – Text containing the name of the storage (optional).

An example of the lake.csv file is presented below:

SUB	LakAMax	LakvA	LakVDead	LakVAmax	LakVMax	LakQMax	LakvQ	NAME
1	0.345501	0.5	345.5	5182.5	5182.5	259.1	1	Storage
2	0.774878	1.5	774.9	11623.2	11623.2	581.2	1	Storage
4	1.1775	1.5	1177.5	17662.5	17662.5	883.1	1	Storage
8	0.344112	1.5	344.1	5161.7	5161.7	258.1	1	Storage
10	0.170432	1.5	170.4	2556.5	2556.5	127.8	1	Storage
11	0.215963	1.5	216	3239.4	3239.4	162	1	Storage
12	0.142023	1.5	142	2130.3	2130.3	106.5	1	Storage
17	0.019627	1.5	19.6	294.4	294.4	14.7	1	Storage
24	0.771532	1.5	771.5	11573	11573	578.6	1	Storage
25	0.632395	1.5	632.4	9485.9	9485.9	474.3	1	Storage
27	0.29937	1.5	299.4	4490.6	4490.6	224.5	1	Storage
28	1.100976	1.5	1101	16514.6	16514.6	825.7	1	Storage
30	0.244616	1.5	244.6	3669.2	3669.2	183.5	1	Storage
31	1.151989	1.5	1152	17279.8	17279.8	864	1	Storage
33	82.68657	1.5	82686.6	1240299	1240299	62014.9	1	Storage
37	1.585266	1.5	1585.3	23779	23779	1188.9	1	Storage

3.3.3 *Land use settings*

Land use settings are restricted to vegetation parameters in LASCAM. Vegetation parameters are defined in six CSV files located in the **vegetation** folder, namely:

- **Grn.csv** – contains information on deep-rooted vegetation fraction;
- **Imp.csv** – contains information on the impervious soil fraction;
- **Max.csv** – contain initial conditions for groundwater elevations;
- **Rip.csv** – contains information on riparian vegetation fraction;
- **Sc.csm** – contains information on leaf area indexes (LAI's); and
- **Sea.csv** – contains information on seasonal distribution of the LAI's.

The file **grn.csv** consists of two columns, one for the sub-catchment ID and the second for the fraction of deep rooted vegetation (%). The first line is a header with the column identifiers with one additional line for each simulated sub-catchment. An example of this file is presented below:

Number	Value
1	14.9
2	6.4
3	16.7
4	0.1
5	50.9
6	4
7	30.3
8	0.1
9	10.4
10	34.4
11	0.2
12	0
13	2.5
14	0

The **imp.csv** file has a similar structure to the **grn.csv** file, with the first column containing the sub-catchment identifier and the second column containing the impervious fraction (%). A column identifier header is followed by one additional line per sub-catchment, as illustrated below:

Catchmen	Value
1	2
2	2
3	2
4	2.7
5	2
6	2
7	2
8	2
9	2
10	2

The files **max.csv** and **rip.csv** have the same formats of **imp.csv** files. The **sc.csv** file contains the spatially distributed values for Leaf Area Index (LAI). It consists of one header line with the column identifier and one additional line for each sub-catchment, containing the following parameters:

- **Catchment** – refers to the sub-catchment ID;
- **GRN** – Leaf Area Index for the deep-rooted vegetation; and
- **RIP** – Leaf Area Index for the riparian vegetation.

An example of the **sc.csv** file is present below:

Catchmen	GRN	RIP
1	1.08	1.08
2	0.81	0.81
3	0.87	0.87
4	0.77	0.77
5	1.9	1.9
6	0.87	0.87
7	0.88	0.88
8	0.92	0.92
9	0.92	0.92
10	0.96	0.96
11	0.74	0.74
12	0.83	0.83

3.3.4 *Rainfall settings*

Rainfall data is defined in three files. The first two files are **rain.csv** and **siteselection.xls** and are located in the folder rainfall. The file **rain.csv** contains the raw observed data from rainfall stations and consists of one header line with one additional line per rainfall record. Inputs required for each rainfall record are:

- **X** – eastern coordinate of the rainfall station;
- **Y** – northern coordinate of the rainfall station;
- **Station_Number** – rainfall station identification number;
- **Date_Time** – date and time of the rainfall record;
- **Year1** – year of the rainfall record;

- **Month1** – month of the rainfall record;
- **Day1** – day of the rainfall record; and
- **Precip** – recorded precipitation in mm.

Rainfall records are required to be in a daily basis. The records must also be ordered by station number and date. An example of the file is included below:

X	Y	Station_Number	Date_Time	Year1	Month1	Day1	Precip
757756.6	6649898	53026	01/01/1990	1990	1	1	0
757756.6	6649898	53026	02/01/1990	1990	1	2	0
757756.6	6649898	53026	03/01/1990	1990	1	3	0
757756.6	6649898	53026	04/01/1990	1990	1	4	0
757756.6	6649898	53026	05/01/1990	1990	1	5	0
757756.6	6649898	53026	06/01/1990	1990	1	6	0
757756.6	6649898	53026	07/01/1990	1990	1	7	0
757756.6	6649898	53026	08/01/1990	1990	1	8	0
757756.6	6649898	53026	09/01/1990	1990	1	9	0
757756.6	6649898	53026	10/01/1990	1990	1	10	0
757756.6	6649898	53026	11/01/1990	1990	1	11	0
757756.6	6649898	53026	12/01/1990	1990	1	12	0
757756.6	6649898	53026	13/01/1990	1990	1	13	0
757756.6	6649898	53026	14/01/1990	1990	1	14	0
757756.6	6649898	53026	15/01/1990	1990	1	15	0
757756.6	6649898	53026	16/01/1990	1990	1	16	0
757756.6	6649898	53026	17/01/1990	1990	1	17	0

The file **siteselection.xls** contains the stations that will be used by LASCAM, if only a sub-set of the rainfall record is planned to be used. This file is redundant if the option **Use User Defined Rain** in the file **runtime.xls** is set to "no".

The format of **site selection.xls** consists of a header line with column identifiers, followed by one line per rainfall station that will be used in model. The followed inputs are required for each station:

- **Site ID** – Rainfall station Id. It must match the **Station_Number** field present in the **rain.csv** file;
- **Y** – northern coordinate of the rainfall station;
- **X** – eastern coordinate of the rainfall station;
- **Projection** – coordinate system used for the rainfall station coordinates

An example of the site selection file is presented in the following figure:

Site ID	Y	X	Projection
53026	6649898	757757	UTM
53034	6665784	723831	UTM
54003	6633786	846923	UTM
55000	6573844	868771	UTM
55006	6494862	806880	UTM
55024	6562915	812045	UTM
55043	6486439	823514	UTM
55045	6546590	788897	UTM
55049	6508161	849470	UTM
55069	6516630	787454	UTM
55136	6565924	915140	UTM
55140	6563356	854517	UTM
55143	6561153	888231	UTM
55176	6534680	889005	UTM
55239	6499982	830297	UTM
55274	6589907	828862	UTM
55276	6578158	833950	UTM
55311	6540048	863834	UTM
56075	6558148	924005	UTM

The file **climate.csv** is present in the climate folder and contains annual averages for rainfall and evaporation for each sub-catchment. The format of the **climate.csv** file is described in the following section.

3.3.5 *Evaporation settings*

Parameters required for evaporation are restricted to a single value of average annual evaporation per sub-catchment. These parameters are set in the **climate.csv** file (same file where average rainfall per catchment is defined) located in the **climate folder**. The file consists of one header line, followed by one line for each sub-catchment, with the following parameters:

- **Link** – represents the sub-catchment ID;
- **Evap** – average annual evaporation (mm);
- **Rai** – average annual rainfall (mm);

The following figure illustrates the **climate.csv** format:

link	Evap	Rai
1	2000	592.96
2	2000	567.7
3	2000	611.3
4	2000	533.35
5	2000	831.23
6	2000	570.51
7	2000	554.82
8	2000	572.2
9	2000	642.43
10	2000	663.65
11	2000	625.02

3.4 Time settings

LASCAM models are simulated in daily time steps. Time settings are restricted to beginning and end of simulation dates. These settings can be accessed and modified in the spreadsheet **runtime.xls** located in the **master** folder.

The fields **StartDate** and **EndDate** (displayed in yellow in the next figure) control the dates for beginning and end of the simulation, respectively. The fields **Start Flow Input** and **End Flow Input** control the period in which observation data will be used for calibration purposes. The use of sub-sets is useful in some situations where an initial simulation period is assigned to the model stabilize numerically to initial conditions before sensible results can be provided.

Runtime				
StartDate	EndDate			
01/01/2006	31/12/2009			
Start Flow Input	End Flow Input			
01/01/1990	01/01/2010			
OutputDirectory				
final_simulation				
Convert Raw Data	Use User Defined Rain	Run Calibrator	Monthly B Store	Run Validation
yes	no	no	yes	yes
Zone				
55H				

3.5 Specific run settings

Other specific run settings can also be found in the **runtime.xls** file and are highlighted in yellow in the screen print below. The specific run settings are:

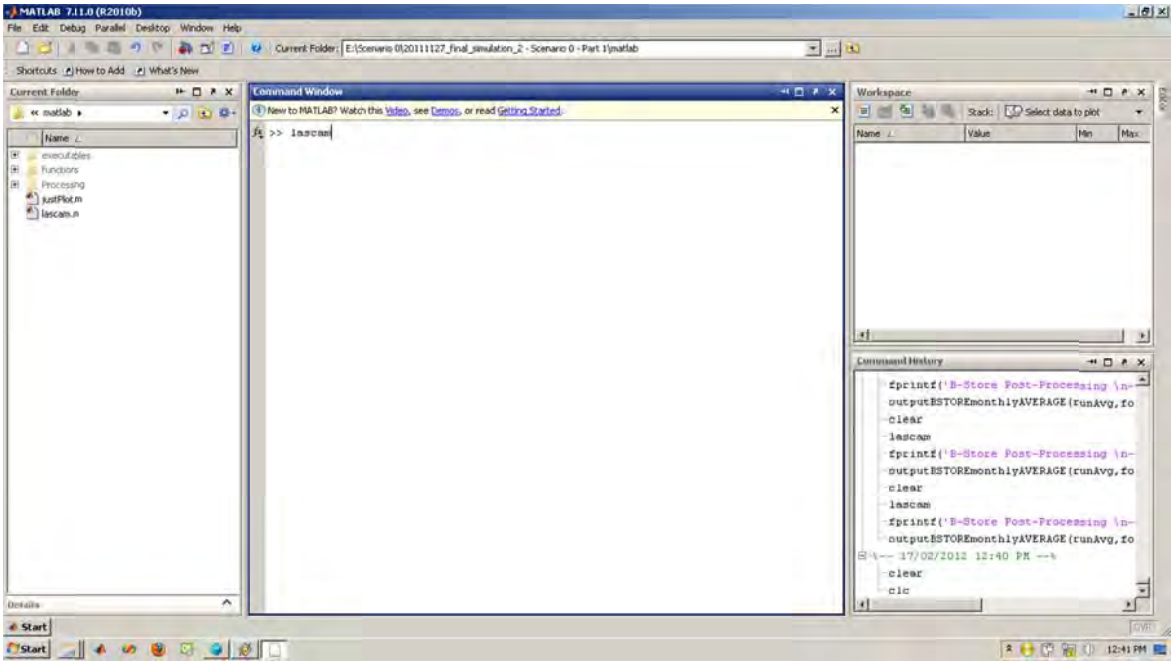
- **OutputDirectory** – defines the name of the output directory which is created (when it does not exist) during the LASCAM run from Matlab;
- **Convert Raw Data** – defines whether the rainfall and observation inputs need to be converted to LASCAM binary format prior to the model run. The raw data conversion can be time consuming depending on the amount of data involved in the simulation, so if the data has been previously converted, it is recommended to set this option to "no";
- **Use User Defined Rain** – defines the rainfall data to be used in the simulation. If this option is set as "no", LASCAM will use the entire rainfall dataset for generation of spatially distributed values. If the option is set as "yes", only the rainfall stations specified in the **siteselection.xls** spreadsheet will be used;
- **Run Calibrator** – defines whether the calibrator is to be used. When set to "no", LASCAM will conduct one single run with the specified parameters. When set to "yes", LASCAM will be run many times and attempt to obtain the best match between the model results and observation data varying the parameters specified in the file **calibration.dat**;
- **Monthly B Store** – defines whether a specific output containing infiltration, evaporation and net balance for each sub-catchment is to be written. If set to "yes" Matlab post-processing will generate a series of charts and spreadsheets for each sub-catchment. If set to "no", output will be ignored; and
- **Run Validation** – defines whether additional plots comparing observed and simulated flow rates are generated.

Runtime				
StartDate	EndDate			
01/01/2006	31/12/2009			
Start Flow Input	End Flow Input			
01/01/1990	01/01/2010			
OutputDirectory				
final_simulation				
Convert Raw Data	Use User Defined Rain	Run Calibrator	Monthly B Store	Run Validation
yes	no	no	yes	yes
Zone				
55H				

3.6 LASCAM runs and post processing results

Once the input files are set, the model can be run from Matlab. Firstly the current folder of Matlab must be set to the **matlab** folder described in section 3.1. To change the current folder, type the folder location in the **Current Folder:** combo box located at the top of Matlab Window, or press the button ... located next to it, as presented in the following figure.

After selecting the current folder, go to the **Command Window** in Matlab, type **lascam** and press Enter. This command starts the Matlab scripts that will translate the input files and call the LASCAM executable. Charts and plots will be displayed showing the observation data as the simulation goes and, once the run is finished, another series of plots will be displayed.



All the plots and spreadsheets generated by Matlab post processing scripts are saved in the subfolder **2.post** located within the **Output** folder. The **2.post** folder contains 3 subfolders, namely:

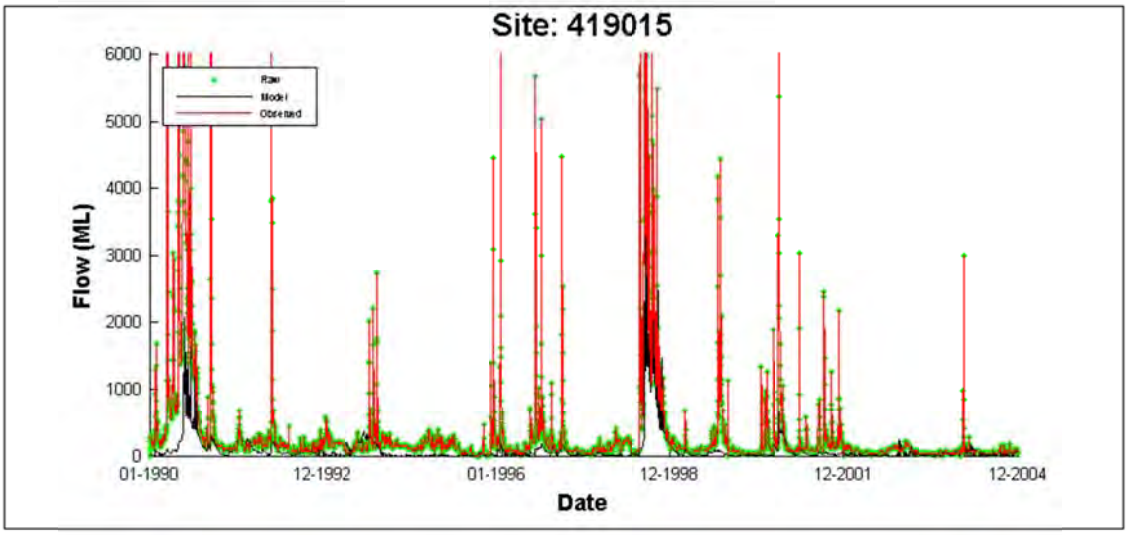
- **BStore;**
- **Flow;** and
- **Validation.**

The **BStore** folder stores the results regarding infiltration and evaporation into deep aquifer, the recharge package file that goes into the Modflow model is saved in this folder. The **CSV** subfolder contains a series of spreadsheets. The **Master_Bstore.xlsx** file contains cumulative monthly values for evaporation and recharge into the aquifer. This file contains the data for all the catchments and entire simulated period. An example of the file is illustrated below.

Subcatchment	Year	Month	Recharge	Evaporation	Recharge - Evaporation
1	1990	1	1.1569	54.5519	-53.395
1	1990	2	1.0381	19.1689	-18.1308
1	1990	3	1.1419	27.8544	-26.7125
1	1990	4	1.0982	1.866	-0.7678
1	1990	5	1.1281	1.8352	-0.7071
1	1990	6	1.0852	0.0117	1.0735
1	1990	7	1.1149	0.0071	1.1078
1	1990	8	1.108	0.0202	1.0878
1	1990	9	1.066	4.8537	-3.7877
1	1990	10	1.0949	36.2512	-35.1563
1	1990	11	1.0533	54.6288	-53.5755
1	1990	12	1.0821	53.9222	-52.8401
1	1991	1	1.0763	30.0574	-28.9811
1	1991	2	0.9673	29.8007	-28.8334
1	1991	3	1.0658	30.0377	-28.9719
1	1991	4	1.0269	15.5213	-14.4944
1	1991	5	1.0559	2.1266	-1.0707
1	1991	6	1.0177	0.0062	1.0115
1	1991	7	1.0467	0.0057	1.041
1	1991	8	1.0424	0.0382	1.0042
1	1991	9	1.004	15.1126	-14.1086
1	1991	10	1.033	51.2204	-50.1874
1	1991	11	0.9949	48.756	-47.7611

In addition to the **Master_Bstore.xlsx** file, additional files containing individual values of recharge and evaporation for each sub catchment are created. The general file name is **scXXX_store.xlsx**, where XXX is the corresponding catchment number. These files have the same format as the master file.

The **Flow** folder contains charts and spreadsheets comparing results from LASCAM against observation data. The subfolder **Plots** contain charts with hydrographs of simulated and observed flow rates, as illustrated below. One plot for each observation point is generated.



The **CSV** folder within **Flow** contains spreadsheets containing the data used to generate the plots. The file **masterFlow.xlsx** contains all the observation data and corresponding LASCAM result values. The spreadsheet contains 5 columns namely:

- **StationID**: contains the name of the observation point;
- **X**: contains the eastern coordinate of the observation;
- **Y**: contain the northern coordinate of the observation;
- **Raw**: contains the observation data (in ML/day);
- **Model**: contains the model results (ML/day);

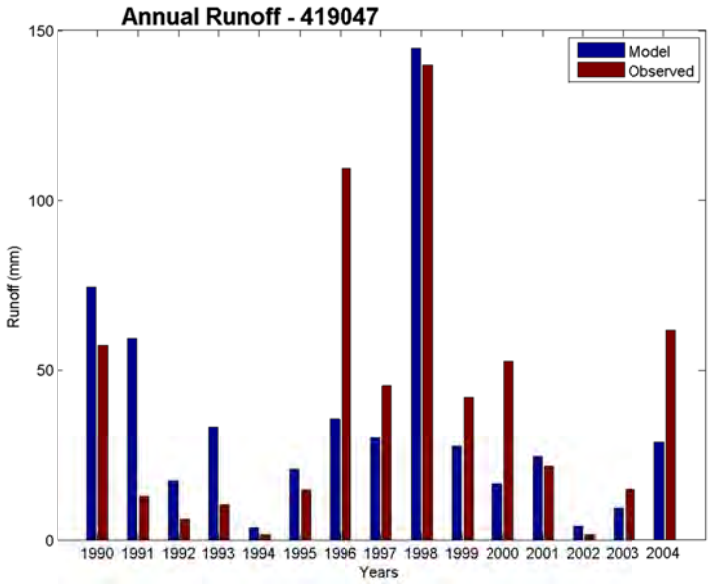
Additional files with the same format are generated individually for each station. The file name corresponds to the station ID. The format of the flow files is illustrated below:

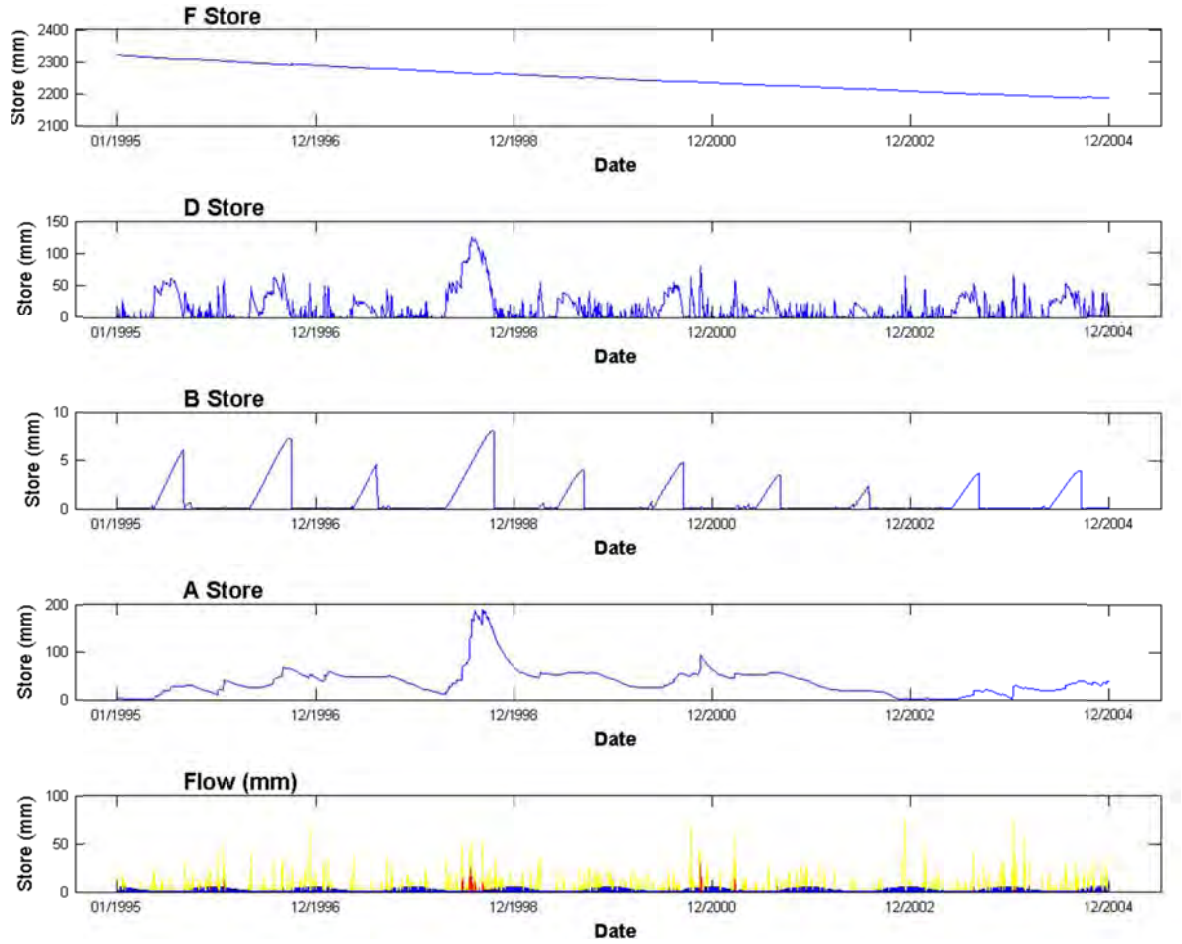
StationID	X	Y	Date	Raw	Model
419015	887628.7	6543187	01/01/1990	0	0
419015	887628.7	6543187	02/01/1990	60.6	0
419015	887628.7	6543187	03/01/1990	127.6	0
419015	887628.7	6543187	04/01/1990	278.2	0
419015	887628.7	6543187	05/01/1990	201.6	0
419015	887628.7	6543187	06/01/1990	226	0
419015	887628.7	6543187	07/01/1990	226	0
419015	887628.7	6543187	08/01/1990	248.6	0
419015	887628.7	6543187	09/01/1990	320	8.1216
419015	887628.7	6543187	10/01/1990	202.4	13.77216
419015	887628.7	6543187	11/01/1990	174.6	22.00608
419015	887628.7	6543187	12/01/1990	184.2	20.89152
419015	887628.7	6543187	13/01/1990	180.6	20.66688
419015	887628.7	6543187	14/01/1990	177.6	19.63872
419015	887628.7	6543187	15/01/1990	172	18.84384
419015	887628.7	6543187	16/01/1990	187.2	18.37728
419015	887628.7	6543187	17/01/1990	198.6	18.46368
419015	887628.7	6543187	18/01/1990	144.2	18.16128
419015	887628.7	6543187	19/01/1990	137	18.13536
419015	887628.7	6543187	20/01/1990	137	17.68608
419015	887628.7	6543187	21/01/1990	141	16.75296
419015	887628.7	6543187	22/01/1990	145	15.84576
419015	887628.7	6543187	23/01/1990	144.6	14.9904
419015	887628.7	6543187	24/01/1990	143.2	0
419015	887628.7	6543187	25/01/1990	161.2	0

The **validation** folder contains additional plots used to assess the consistency of model results at catchments where observations are present. The subfolder **Plots** contains the following plots:

- **Annual Runoff** – which compares simulated and observed runoff values in an yearly basis;
- **Cumulative Total** – plotting cumulative observed and simulated runoff rates;
- **Flow Duration** – which compares simulated and observed values of runoff against percentage of time equalled or exceeded;
- **Monthly Average** – which plot monthly averaged values for observed and simulated runoff rates;
- **Monthly Flow** – which plot monthly simulated runoff values against corresponding monthly averaged observations;
- **Monthly Average Recharge** – which presents monthly average infiltration values (mm).
- **Stores** – which present time series of storage volumes for the A, B, D and F stores (as defined in LASCAM) in millimetres;

These 7 plots are generated for each observation station. The files are names as mentioned above, with the station name append at the end. Illustrative examples of the **Annual Runoff** and **Stores** plot are presented below.




F Store
Store (mm)
2400
2300
2200
2100
01/1995
12/1996
12/1998
12/2000
12/2002
12/2004
Date
D Store
Store (mm)
150
100
50
0
01/1995
12/1996
12/1998
12/2000
12/2002
12/2004
Date
B Store
Store (mm)
10
5
0
01/1995
12/1996
12/1998
12/2000
12/2002
12/2004
Date
A Store
Store (mm)
200
100
0
01/1995
12/1996
12/1998
12/2000
12/2002
12/2004
Date
Flow (mm)
Store (mm)
100
50
0
01/1995
12/1996
12/1998
12/2000
12/2002
12/2004
Date

THIS PAGE HAS BEEN LEFT BLANK INTENTIONALLY

REFERENCES

ESI 2011, Guide to Using Groundwater Vistas Version 6. Environmental Simulations, Inc.

Harbaugh, AW, Banta, ER, Hill, MC & McDonald, MG 2000, MODFLOW-2000, The U.S. geological survey modular ground-water model – User guide to modularisation concepts and the ground-water flow process, Open- File Report 00-92, pp. 121 United States Geological Survey, USA.

Viney, NR, & Sivapalan, M 2000, Modelling catchment processes in the Swan-Avon River Basin. *Hydrol. Process., 15*(13), 2671-2685.